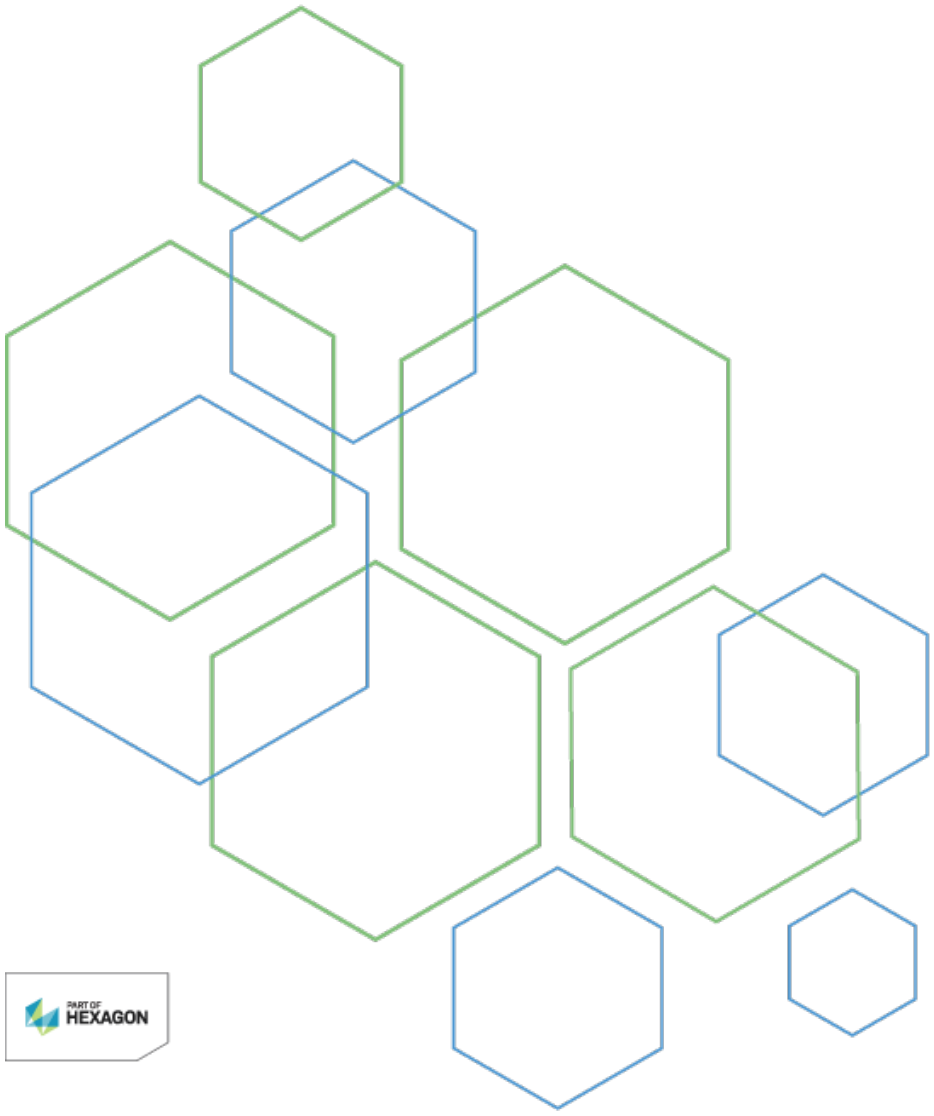
INTERGRAPH Smart 3D

Compartmentation

User's Guide



PART OF HEXAGON

Version 2016 (11.0)
November 2016

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Contents

Preface

This document is a user's guide for the Compartmentation functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Compartmentation

The following changes have been made to the Compartmentation task.

Version 2016 (11.0)

- Added the **Submit Batch Job** command to run batch processes using the Intergraph Batch Services framework. For more information, see *Submit Batch Job* (on page 120). (P2 CP:273093, P2 CP:178040)
- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 14). (P2 CP:271166)
- Added the **Assign Design Parent** command to specify the design and space parents for compartments and volumes. For more information, see *Assign Design Parent to Compartments and Volumes* (on page 111). (P2 CP:278727)
- Renamed the **Compartmentation Query Service** command to **Query Service**. For more information, see *Query Service Command* (on page 112). (P3 CP:285197)
- The **Delete Optional** property in **Model Data Transform** now supports compartmentation entities. (P2 CP:263606)
- Added descriptions for the query types in **Query Service**. For more information, see *Query Service Dialog Box* (on page 116). (P4 CP:289946)

SECTION 1

Compartmentation

The **Compartmentation** task in Intergraph Smart™ 3D allows you to define the functional layout of a ship by creating compartments, ship zones, and other volume objects within the ship. Using the **Compartmentation** task, you can also assign loads and load combinations to volume objects.

Compartments are physical spaces within the ship that cannot overlap, except for some limited cases where one compartment, such as a fuel oil day tank, is completely enclosed by another compartment (such as the main machinery space). In almost all instances, a compartment is completely defined in terms of topological boundary relationships with a plate system, a deck, a transverse bulkhead, a longitudinal bulkhead, or a hull shell object.

Ship zones are logical spaces within and around a ship that may or may not physically overlap. In most cases, ship zones are defined by a rectangular bounding box. The properties and specifications of compartments, ship zones, and other volumes are defined in the reference data in the Catalog task.

You can define the geometry of a volume object in a variety of ways - by two points, by four points, by other surfaces, or by other volumes. You can also create a volume object in the model by defining a path and projecting a cross-section along the path. In addition to these functions, you can import a SAT file to create compartments.

After creating volumes, you can specify loads that are applied to the volumes. Load folders, spatial loads, unit loads, and load combinations appear on the **Analysis** tab in the **Workspace Explorer**.

You can start the Compartmentation task by clicking **Tasks > Compartmentation**. The Compartmentation task has the following commands:

Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 14).

Create Space Folder - Creates a new folder for the organization of volume objects in the model. For more information, see *Create Space Folder* (on page 16).

Place Volume by Two Points - Defines a volume by selecting two points to represent opposite vertices of the volume. For more information, see *Place Volume by Two Points* (on page 22).

Place Volume by Four Points - Defines a volume by selecting three points to define a base plane and a fourth point to define elevation or depth. For more information, see *Place Volume by Four Points* (on page 31).

Create Imported Volume - Imports an ACIS file (.sat) into the model to create volumes. For more information, see *Create Imported Volume* (on page 45).

Place Volume Along Path - Places a volume by specifying a cross-section and projecting the cross-section along a path. For more information, see *Place Volume Along Path* (on page 47).

Create Volume By Faces - Defines a volume by selecting sufficient bounding surfaces to create a closed volume. For more information, see *Create Volume by Faces* (on page 69).

Create Multiple Volumes - Creates multiple compartments in a specified volume. For more information, see *Create Multiple Volumes* (on page 76).

Merge Volumes - Merges multiple existing volumes into a single new volume. For more information, see *Merge Volumes* (on page 86).

Bound by Volumes - Defines a volume from sufficient bounding spaces that create a closed volume. For more information, see *Bound by Volumes* (on page 90).

Place Volume by Window - Defines a volume based on the active graphical view of the model. The view boundaries, depth range, and possible clipping are used when generating the volume. For more information, see *Place Volume by Window* (on page 34).

Place Volume by Selectset - Defines a rectangular volume that encloses a selected set of objects in the model. For more information, see *Place Volume by Selectset* (on page 40).

Create Load Folder - Creates a new folder for the organization of load objects. For more information, see *Create Load Folder* (on page 94).

Create Load Combination - Combines spatial and unit loads. For more information, see *Create Load Combination* (on page 96).

Create Spatial Load - Creates a load on a spatial volume. For more information, see *Create Spatial Load* (on page 100).

Create Unit Load - Creates a load on a point. For more information, see *Create Unit Load* (on page 102).

Export Compartment - Exports a compartment solid body as a .sat file. This command is on the **File** menu. For more information, see *Export Compartment* (on page 109).

Create Compartments Automatically - Creates compartments through an entire ship in one operation. This command is located on the **Tools** menu. For more information, see *Create Compartments Automatically* (on page 81).

Update Attributes - Updates the attributes of selected compartments based on custom code. This command is located on the **Tools** menu. For more information, see *Update Attributes* (on page 108).

Submit Batch Job - Runs batch processes using the Intergraph Batch Services framework. This command is available on the **Tools** menu. For more information, see *Submit Batch Job* (on page 120).

In addition to commands on the vertical toolbar and menus, the Compartmentation task also provides a custom command, **Compartment Query Service**, which allows you to perform queries on objects in the model, for example, to find all compartments that are on or cross a deck. For more information, see *Query Service Command* (on page 112).

SECTION 2

Compartmentation Workflow

A compartment is a type of volume object representing a physical space, such as a cabin, a machinery room, or a ballast tank, in a marine model. Smart 3D places all compartments in the model using information defined in the compartmentation reference data. Using the reference data workbook, you can create custom compartments that fit your specific requirements. Your first step should be to review, edit, and otherwise customize the delivered compartmentation reference data. For more information about customizing the reference data that is delivered with the Compartmentation task, refer to the *Compartmentation Reference Data Guide*, available from the **Help > Printable Guides** command.

After the reference data is customized, go to the Systems and Specifications task and begin to define the systems that you want in your model. While you are not absolutely required to create your systems first, doing so keeps you from having to edit your volume objects after placement to assign them to the correct system.

With the systems defined, use the Grids task to define and place coordinate systems, elevation planes, and grid planes. Then, use the Molded Forms task to define the basic structure of the ship, including: the hull, decks, bulkheads, major openings, stiffeners, beams, design seams, and so forth.

After the compartmentation reference data and the needed systems, grids, planes, and ship structures are defined, you can begin creating compartments, ship zones, and other volume objects in your model.

When the volume objects are defined, you can create load objects, including spatial loads, unit loads, and load combinations.

Compartmentation Common Tasks

The following tasks are used frequently in the Compartmentation task.

Customize Reference Data

Create new volume object types by editing the compartmentation workbooks. For more information, see the *Compartmentation Reference Data Guide*, available from the **Help > Printable Guides** command.

Organize Volumes and Loads

Create folders to organize volume and load objects. For more information, see *Create a space folder* (on page 17) and *Create a load folder* (on page 95).

Create Volume Objects

Create volume objects in the model. There are several methods of creating volumes in the Compartmentation task. You can place a volume by two or four points, import volumes, place a volume along a path, create a volume bound by faces, create multiple volumes at once, or merge volumes.

- *Place a volume by two points* (on page 23)
- *Place a volume by four points* (on page 32)
- *Import a volume from an ACIS or IGES file* (on page 46)
- *Place a volume along a path* (on page 57)
- *Place a volume bound by faces* (on page 72)
- *Create multiple volumes in part of a ship* (on page 78)
- *Create multiple volumes in an entire ship* (on page 82)
- *Merge multiple volumes* (on page 88)
- *Place a volume bound by volumes* (on page 92)

Create Loads

Create loads associated with the volumes in the model. You can create load combinations, spatial loads, and unit loads.

- *Create a load combination* (on page 97)
- *Create a spatial load* (on page 101)
- *Create a unit load* (on page 103)

SECTION 3

Selecting Objects

All volume objects in the Compartmentation task have properties that you can edit. Using **Select** on the vertical toolbar, you select the object that you want to edit.

An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view or in the **Workspace Explorer**. For example, if you select **Compartment**, you can select only compartments in a graphic view or in the **Workspace Explorer**.

The Compartmentation task includes these filters:

All Folders

Limits your selection in the **Workspace Explorer** to folders.

All Load Objects

Limits your selection in a graphic view or in the **Workspace Explorer** to load objects.

All Volumes

Allows you to select any volume object in a graphic view or in the **Workspace Explorer**.

Compartment

Limits your selection in a graphic view or in the **Workspace Explorer** to volume objects that are defined as compartments.

Construction Graphics

Limits the selection of items to construction graphics.

Control Points

Limits your selection in a graphic view or in the **Workspace Explorer** to control points created using **Insert** > **Control Point**.

Interference Zones

Limits your selection in a graphic view or in the **Workspace Explorer** to volumes defined as interference zones.

Load Combination

Limits your selection in a graphic view or in the **Workspace Explorer** to load combinations.

Load Folder

Limits your selection in the **Workspace Explorer** to load folders.

Region

Limits your selection in a graphic view or in the **Workspace Explorer** to volumes defined as regions.

Ship Zone

Limits your selection in a graphic view or in the **Workspace Explorer** to volume objects that are defined as zones.

Space Folder

Limits your selection in the **Workspace Explorer** to space folders.

Spatial Load

Limits your selection in a graphic view or in the **Workspace Explorer** to spatial loads.

Unit Load

Limits your selection in a graphic view or in the **Workspace Explorer** to unit loads.

Void Space

Limits your selection in a graphic view or in the **Workspace Explorer** to volume objects that are defined as void spaces.

All

Allows you to select any object, even objects created in another task.

 **Inside Fence**

Limits your selection to filtered objects entirely inside the fence.

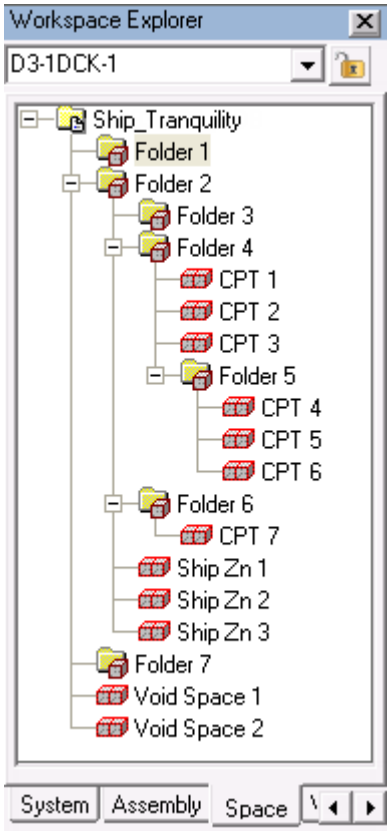
 **Overlapping Fence**

Limits your selection to filtered objects entirely inside the fence and those objects outside but touching the fence.

SECTION 4

Create Space Folder

Creates a new hierarchical folder for organizing your volume objects. You can create folders to provide a hierarchical storage system for the volume objects that you create in the Compartmentation task. When you select the **Create Space Folder** command, a ribbon appears that allows you to define a parent and name for the new folder. After the space folder is created, the space folder icon appears on the **Space** tab in the **Workspace Explorer**.



The **Space** tab of the **Workspace Explorer** displays all the volumes in the workspace in a classification hierarchy that reflects the various relationships defined for the volume objects. The content represents the current volume objects loaded from the database into the active workspace. When you select a volume object in the **Workspace Explorer**, it appears highlighted with the select color in the graphic view. Additionally, the software defaults to edit mode and displays the **Edit Volume** ribbon, which you can use to modify the selected volume object.

TIP You can also right-click a folder on the **Space** tab of the **Workspace Explorer**, and then select **Create Compartment Folder**.

Create Space Folder Ribbon

Sets options for creating a new space folder. After you have completed the required fields, you can click the **Space** tab in the **Workspace Explorer** to view the results.

Properties

Displays the **Space Folder Properties** dialog box, which allows you to set properties for the folder that you are creating. For more information, see *Space Folder Properties Dialog Box* (on page 154).

Finish

Creates the space folder with the **Name** and **Parent** properties that you specify.

Name

Specifies a name for the new space folder.

Parent

Specifies a location within the space hierarchy. The space folder becomes a child of the selected parent. Selecting **More** in the dropdown list opens the **Select Space Folder** dialog box, which displays the complete space hierarchy.

What do you want to do?

- *Create a space folder* (on page 17)

Create a space folder

1. Click **Create Space Folder** on the vertical toolbar.

   TIP You can also right-click a folder on the **Space** tab of the **Workspace Explorer**, and then select **Create Compartment Folder**.

2. On the ribbon, type a name for the new space folder.
3. Specify a parent for the new space folder in the **Parent** list.

   TIP You can select **More** in the drop-down list to display the *Select Space Folder Dialog Box* (on page 17), which displays the full hierarchy.

4. Click **Finish**.

NOTE You can click **Properties** on the ribbon to modify the folder properties. For more information, see *Space Folder Properties Dialog Box* (on page 154).

Select Space Folder Dialog Box

Specifies the parent needed for determining placement of a new folder within the space hierarchy. This dialog box appears when you select **More** in the **Parent** box on the **Create Space Folder** ribbon. By browsing through the space hierarchy, you can select a parent for the new space folder. After you select the parent, the software returns you to the model, where you can finalize the creation of the new folder.

Look In

Specifies from which location you want the software to pull hierarchical information. You can retrieve hierarchical information from either the current workspace or from the entire Model database.

See Also

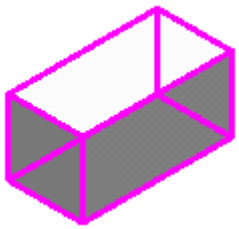
Create Space Folder (on page 16)

SECTION 5

Create Volume Objects

Several methods of creating new volume objects exist in the Compartmentation task. You can provide two or four points to identify a rectangular cube, or you can select existing boundaries, such as decks or bulkheads, to define the volume. You can also import volumes or place a volume along a path. It is up to the designer to determine which volume creation tools are most applicable for the specific requirements of a ship.

Volumes are stored in the model. In a graphic view, a volume object appears as a gray box, by default.



In the **Workspace Explorer**, a volume is represented by an icon on the **Space** tab. You can locate and manipulate volumes quickly and easily in the space hierarchy.

You can use SmartSketch points when you create volume objects. When you associate a set of objects to a volume point with SmartSketch relationship indicators, if the associated objects change, then the volumes update accordingly. For example, if a volume is connected to a grid intersection or a reference plane and the grid or plane is moved, the associated volume is added to the To Do list, and you can update the volume size.

Another example is when the boundaries of volumes are defined by plate systems. When the structural designers move the plate systems that are used to define compartments, the software places the volumes on the To Do list for updates.

NOTE Most volumes created in this task do not update automatically, and therefore, must be manually updated. You should check the To Do list regularly for out-of-date volumes, especially in a concurrent design environment.

Merging or Splitting Volumes

After you create a volume, you can add to or subtract from the volume using Boolean operations. You can merge the volume with other volumes. You can also split the volume into multiple volumes. When you perform these operations, the software changes the type for the input volume(s) to a void space if the input volume has the same type as the output volume(s).

For example, when you click the **Create Multiple Volumes** command, you select an existing volume and then select the inner faces. The existing, parent volume is not deleted, but it is changed to a void space if the parent is the same type as the children volumes.

Creating Volumes by Points

When doing the preliminary design of compartments and ship zones, it is often convenient to quickly create volumes by two or four points. Using the **Place Volume By Two Points**

command and **Place Volume By Four Points** command, you select points in the model to define a volume.

For the **Two Points** command, you select points (bottom-front-left and top-right-back points) at the corners of the space. For the **Four Points** command, you first select three points to define the base planar surface of the volume, and then you select a point to provide the necessary elevation to extrapolate the plane in three dimensions.

Alternatively, you can type the X-, Y-, and Z-coordinates of the points using the **PinPoint** command. Either way, these volumes are not linked or associative to any existing geometry. Only their location coordinates are relative to the defined origin of the model.

You can also use the Boolean operation buttons on the ribbon to add or subtract volumes from the overall volume.

For more information about creating a volume by two points, see *Place Volume by Two Points* (on page 22); for information about creating a volume by four points, see *Place Volume by Four Points* (on page 31).

Creating Imported Volumes

The Compartmentation task allows you to quickly create volumes by importing an ACIS file into the model. For more information, see *Create Imported Volume* (on page 45).

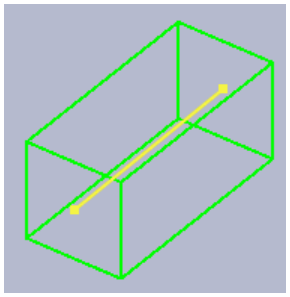
NOTE This functionality is similar to the **Import Plate System** command in the Molded Forms task. For more information, see the *Molded Forms User's Guide*, available with the **Help > Printable Guides** command.

Placing Volumes along Paths

In the Compartmentation task, you can create customized volumes in the location that you need and with the shape that you need using the **Place Volume Along Path** command. This command is especially useful for volumes that need to have an unusual shape or that must negotiate around specific objects in the model. For example, if you need to leave adequate space for equipment to reach a particular location, the **Place Volume Along Path** command allows you to specify the path where the equipment needs to travel as well as the appropriate amount of clearance that must be left on all sides. One application of this command is a fire evacuation route.

When you place a volume along a path, a two-dimensional cross section is projected along the path that you specify in the model to create the volume. The path determines the location of the volume in the model. The cross section, on the other hand, defines the shape and dimensions of the volume.

The following picture shows a path (yellow) and a rectangular cross section projected along that path, with the resulting volume shown in green.



Defining Paths

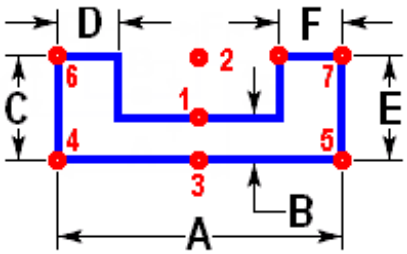
When you define the path along which a cross section is projected, you can choose from a straight line or an arc by three points. Alternatively, if you want to break the path at a particular point, you can select **No Line**. You can also control characteristics of the path by specifying the types of turns (bend, cornice, or chamfer), the dimensions of the turns, and the plane for the path.

Defining Cross Sections

When you define the cross section for the volume, you can select from a standard set of cross sections defined in the reference data, or you can sketch your own two-dimensional cross section.

When you use a standard cross-sectional type, you can change the dimensions and the cardinal point of the cross section on the **Cross-Section** tab on the **Volume Properties** dialog box. The cardinal point of the cross section is important in determining the shape and location of the resulting volume. The cardinal point is the point where you want the software to attach the cross section to the path. All standard cross section types have cardinal points from which you can choose.

For example, in the following picture, you can see that the cardinal point (represented by a black dot) is located in the center of the cross section. If the cardinal point were moved to another location, the resulting volume would be different.



When you sketch a cross section, you must sketch the cross section on the two-dimensional plane that is orthogonal to the first leg of the path. The software displays this plane, which is perpendicular to the path, as you sketch the cross section. The cardinal point is also displayed as you sketch.

Editing Customized Volumes

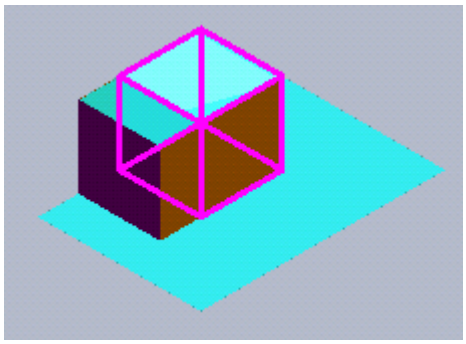
After creating customized volumes, you can edit them. The edit path ribbons allow you to modify the arcs, turns, or straight segments that are part of an existing path, either at placement or afterward.

For more information about creating and editing customized volumes, see *Place Volume Along Path* (on page 47).

Creating Volumes by Faces

Another way to create a volume is by selecting plate systems or reference planes in the model. For more information, see *Create Volume by Faces* (on page 69).

While using the **Create Volume by Faces** command, you select the faces for the compartment. The software determines if there is any ambiguity and if so, allows you to choose among the resulting volumes.



When you are creating a volume by faces, it is often helpful to select a specific coordinate system (grid system). The **Plate Filter** box on the ribbon allows you to select a grid system for this purpose. For more information, see *Create a filter for a grid system* (on page 80).

See Also

Create Multiple Volumes Simultaneously (on page 76)
Create Volume Objects from Existing Volumes (on page 84)

Place Volume by Two Points

Defines a volume by selecting two points to represent the opposite vertices of the volume that you need. You can locate two points using standard tools, such as PinPoint , or by using SmartSketch relationship indicators.

While using this command, you can associate the volume with a design specification from the Catalog database. You can also add or subtract volumes to or from the overall volume.

In a graphic view, a volume appears as a gray box, by default. In the **Workspace Explorer**, a volume is represented by an icon on the **Space** tab.

Place Volume by Two Points Ribbon

Sets options for placing volume objects defined by opposite vertices using the **Place Volume by Two Points** command.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Volume Point 1

Defines the first point for the definition of the volume.

Volume Point 2

Defines the second point for the definition of the volume.

Handles

Allows you to drag the sides of the volume to resize it. This option only displays when you modify a volume.

Select Volumes for Addition

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

Select Volumes for Subtraction

Allows you to select volumes (graphically or from the **Workspace Explorer**) and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28) dialog box, which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Finish

Places the volume as specified. This option only appears when you modify a volume.

Name

Displays the default name for the volume object (as dictated by the active name rule), and allows you to type a different name, if needed. Names must be unique across the entire model.

Type

Specifies the type of volume. Selecting **More** opens the *Select Volume Dialog Box* (on page 29) dialog box from which you can select a type.

Space Folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

What do you want to do?

- *Place a volume by two points* (on page 23)
- *Place a volume by two points using PinPoint* (on page 24)
- *Modify a point defining a two point volume* (on page 27)
- *Modify the face of a two point volume* (on page 28)

Place a volume by two points

1. Click **Place Volume by Two Points** on the vertical toolbar.

   *The **Place Volume by Two Points** ribbon displays.*
2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

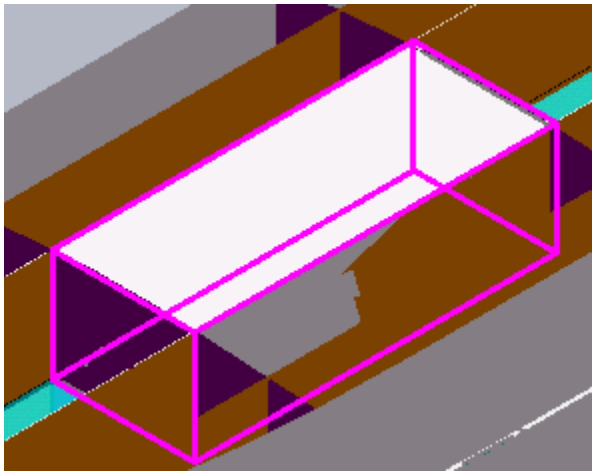
   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. In the graphic view, select the first point of the rectangular volume, and then select the opposite corner of the rectangular volume.

   TIP You can use SmartSketch relationship indicators, such as intersection and point on surface, to assist in positioning the corners of the volume. For more information on the available relationship indicators, see *SmartSketch Options* in the *Common User's Guide*.

   The software creates the volume.

6. Optionally, click **Select Volumes for Addition** or **Select Volumes for Subtraction** on the ribbon to add to or subtract other existing volumes from the resultant volume.

NOTES

- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

Place a volume by two points using PinPoint

1. If the **PinPoint** ribbon is not visible, select **PinPoint** .

   *The **PinPoint** ribbon displays.*

2. On the **PinPoint** ribbon, click **Rectangular Coordinates** .

3. Click **Place Volume by Two Points** on the vertical toolbar.

   *The **Place Volume by Two Points** ribbon displays.*

4. Specify a name for the volume in the **Name** box.
5. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

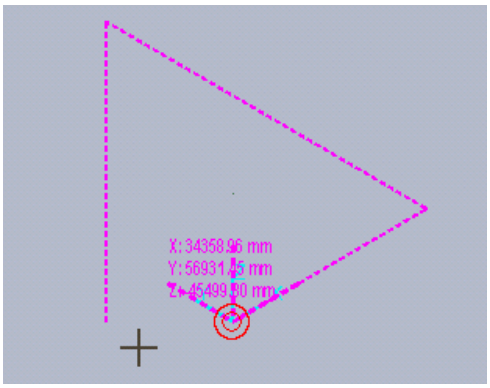
6. Select a folder from the **Space folder** list.

   TIPS

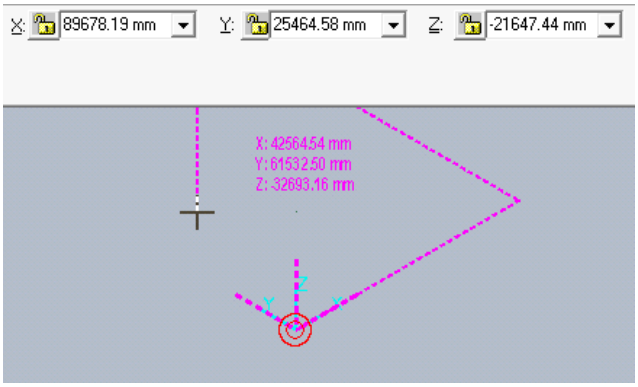
   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

7. Define the first point of the rectangular volume, using one of the following methods:

   - In the graphic view, select the first point, using the PinPoint dashed line axes and coordinate display as a guide.

   - On the **PinPoint** ribbon, type **X, Y**,and **Z** coordinate values. Click **Lock/Unlock** to lock the value for each coordinate. After all coordinate values are locked, click in the graphic view.

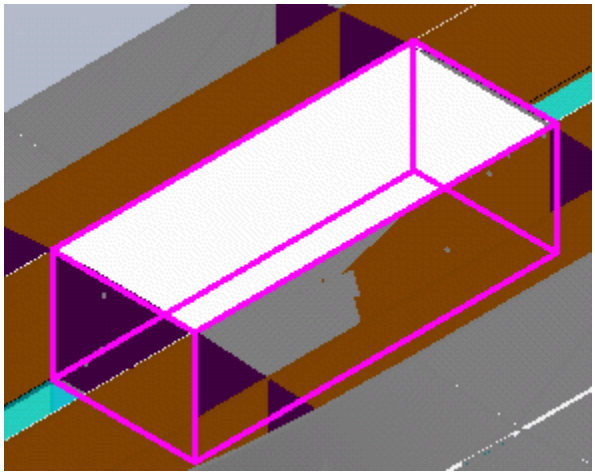
     

TIPS

- By default, the software displays help lines and distance values as you create the volume. You can use **Display On/Off** on the **PinPoint** ribbon to toggle the display on and off.
- PinPoint point coordinate values are relative to a target location. You can use **Reposition Target** to select a new target location.
- For more information on using **PinPoint**, see *PinPoint* and *Place objects using rectangular coordinates* in the *Common User's Guide*.
- You can also use SmartSketch relationship indicators such as intersection and point on surface to assist in positioning the corners of the volume. For more information on the available relationship indicators, see *SmartSketch Options* in the *Common User's Guide*.
- You can also click the middle mouse button or wheel to lock the coordinate values.

8. Define the second point of the rectangular volume using one of the same methods from the previous step.

   The software creates the volume.



9. If necessary, click **Select Volumes for Addition** or **Select Volumes for Subtraction** on the ribbon to add to or subtract other existing volumes from the resultant volume.

NOTES

- By default, the global coordinate system is used to orient the rectangular volume relative to the placement points; however, you can use the **Coordinate System** list on the **PinPoint** ribbon prior to placement to select a different active coordinate system. A placed volume object maintains a local coordinate system that is used in future edits of the volume object. The orientation of the volume object does not change if the active coordinate system changes.
- If a different coordinate system is selected in the **Coordinate System** field, you must click **Set Target to Origin** (in plant mode) or **Set Target to Origin** (in marine mode) to reset the ribbon values.
- You can use function keys with the **PinPoint** command.

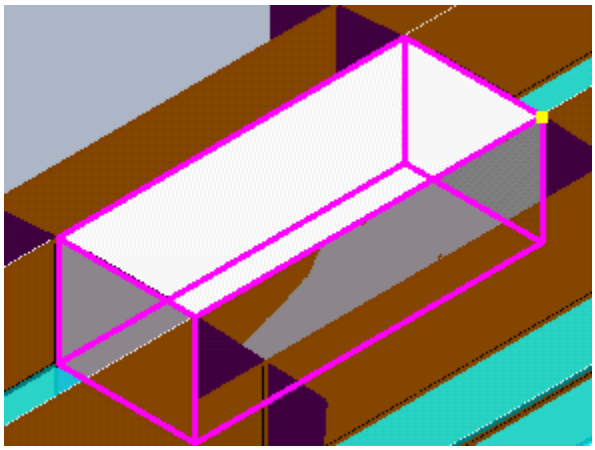
- In plant mode, press **F6** to lock and unlock the E value or absolute distance. In marine mode, press **F6** to lock and unlock the X value or absolute distance.
- In plant mode, press **F7** to lock and unlock the N value or horizontal angle. In marine mode, press **F7** to lock and unlock the Y value or horizontal angle.
- In plant mode, press **F8** to lock and unlock the EL value or vertical angle. In marine mode, press **F8** to lock and unlock the Z value or vertical angle.
- Press **F9** to turn the display of help lines and distance values on and off.
- Press **F12** to reposition the target.

Modify a point defining a two point volume

1. On the vertical toolbar, click **Select** .
2. Select the volume to modify.

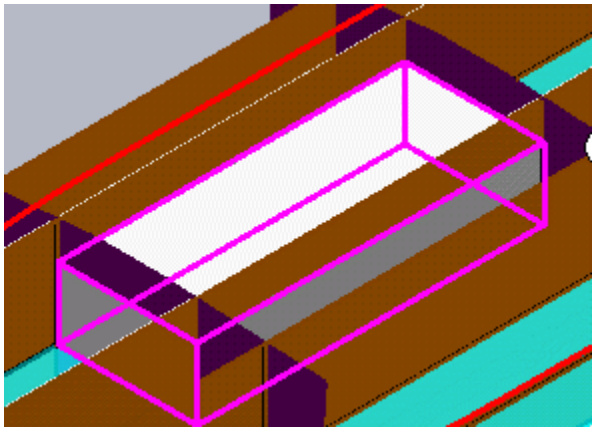
   *The volume highlights, and the **Modify** ribbon displays.*
3. Click **Volume Point 1** or **Volume Point 2** .

   The point displays in the graphic view.



4. Select a new location for the point.

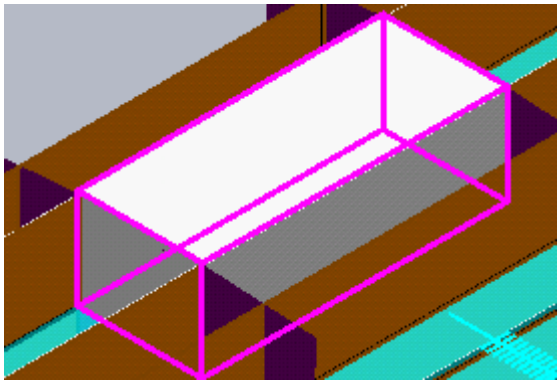
   The software modifies the volume.



Modify the face of a two point volume

1. On the vertical toolbar, click **Select** .
2. Select the volume you want to modify.

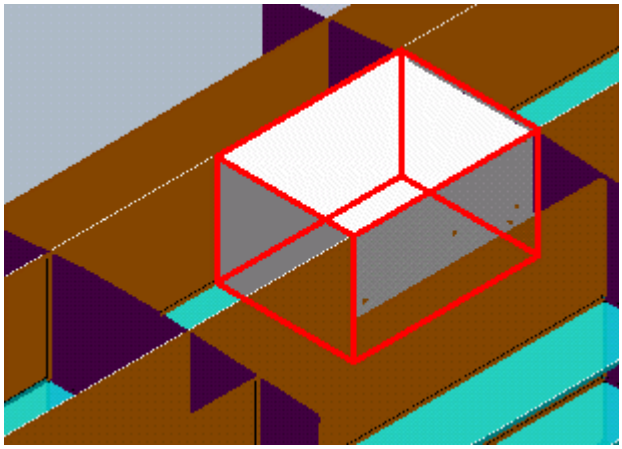
   *The volume highlights and the **Modify** ribbon displays.*



3. Click **Handles** .
4. Select the face of the volume to modify.
5. If a point defining the volume is constrained, such as at an intersection, a dialog box displays explaining the existence of the constraints. Click **Yes** to remove the constraint.
6. Drag the volume face to the new location.

   TIP Use **QuickPick** to assist in selection of the required side.

   The software modifies the volume.



Boolean Operation List Dialog Box

Lists the volumes that have been added to or subtracted from the overall volume definition. You can click **Add** to add a volume or **Remove** to remove a volume from the definition. The order of the volumes in the list does not affect the output.

Added Volumes Tab (on page 29)
Subtracted Volumes Tab (on page 29)

See Also

Create Volume Objects (on page 19)
Compartmentation (on page 9)

Added Volumes Tab

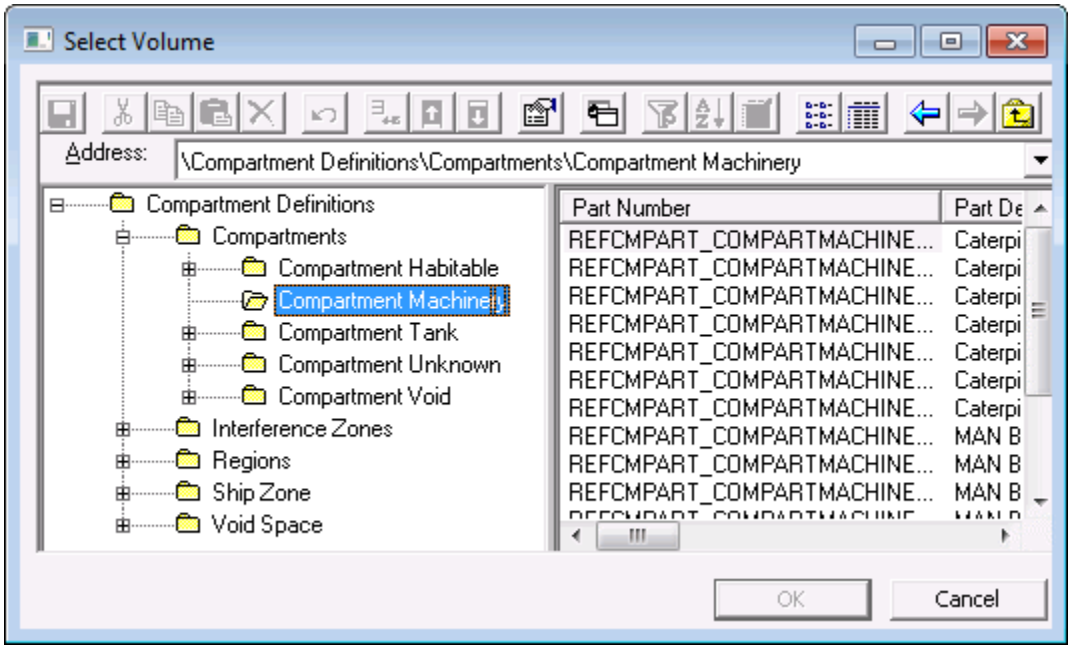
Lists the volumes that have been added to the overall volume definition.

Subtracted Volumes Tab

Lists the volumes that have been subtracted from the overall volume definition.

Select Volume Dialog Box

Specifies the type of volume needed for placement. This dialog box appears when you select **More** in the **Type** box on a horizontal ribbon. By browsing through the space hierarchy, you can find any type of volume defined in the catalog database. After you select a volume object, the software returns you immediately to the model, where you can finalize configuration and placement of the volume.



Save - Saves the active row to the catalog database. This command is available only in the **Catalog** task.

Copy - Copies the selected object. This command is available only in the **Catalog** task.

Paste - Pastes a copied object. This command is available only in the **Catalog** task.

Delete - Deletes the selected object. This command is available only in the **Catalog** task.

Undo - Reverses the most recent operation. This command is available only in the **Catalog** task.

Insert Row - Inserts a blank row into the grid view. This command is available only in the **Catalog** task.

Move Up - Moves the select list entry up one in the editable grid. This command is available only in the **Catalog** task.

Move Down - Moves the select list entry down one in the editable grid. This command is available only in the **Catalog** task.

Properties - Displays the properties of the selected object. The properties on this dialog box are read-only.

Preview - Opens a bitmap file that was assigned to a part or part class in the reference data.

Filter - Filters the data in the content view to quickly find what you are looking for. You must first switch to **Grid View** to make the **Filter** button available.

Sort - Sorts data in the content view by multiple columns to quickly find what you are looking for. This command is available only in the **Catalog** task.

Customize Current View - Controls which property columns display in the content view and in what order. This command is available only in the **Catalog** task.

List View - Displays the information in the content view in a list format.

Grid View - Displays the information in the content view in a table format.

Back - Highlights the last selected item or folder.

Forward - Advances the display forward to return the browser display to the last item that you selected before you used the **Back** command.

Up One Level - Moves the focus up one level in the catalog hierarchy.

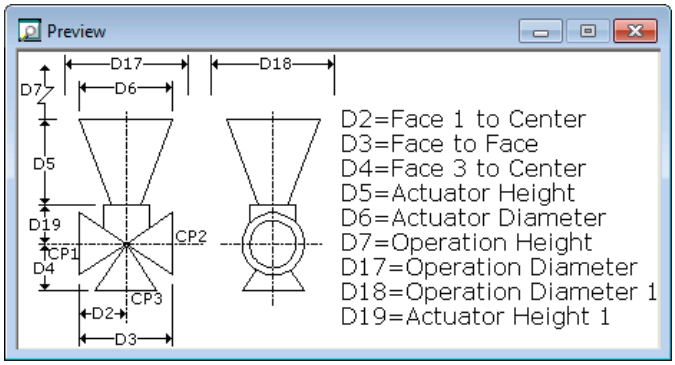
Address - Specifies your exact location within the displayed hierarchy.

See Also

Create Volume Objects (on page 19)
Compartmentation (on page 9)

Preview Dialog Box

Displays bitmap image associated with the selected object. You can leave this **Preview** window open as you select different part and part classes.



Preview graphics must be added to the reference data before you can view the graphics in the catalog. For more information on defining a graphical preview of a specific part, see "Add a Preview Graphic to Parts" in the *Smart 3D Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Place Volume by Four Points

 Defines a volume by selecting three points to define a base plane and a fourth point to define the elevation or depth. While using this command, you can associate the volume with a design specification from the catalog database. You can also add or subtract volumes to or from the overall volume.

By selecting the three planar points in a plan view (looking down), you can ensure that the volume is parallel to the base elevation plane of your design.

Place Volume by Four Points Ribbon

Sets options for placing a volume defined by three planar points and an elevation point.

 **Volume Properties**

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

 **Volume Point 1**

Defines the first point for the definition of the volume.

 **Volume Point 2**

Defines the second point for the definition of the volume. This point determines the length of the volume in one direction.

 **Volume Point 3**

Defines the third point for the volume. This point determines the width of the volume and defines the horizontal plane of one face.

 **Volume Point 4**

Defines the elevation, or depth, of the new volume.

 **Handles**

Allows you to drag the sides of the volume to resize it. This option only displays when you modify a volume.

 **Select Volumes for Addition**

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

 **Select Volumes for Subtraction**

Allows you to select volumes (graphically or from the **Workspace Explorer**) and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Finish

Places the volume as specified. This option only appears when you modify a volume.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

What do you want to do?

- *Place a volume by four points* (on page 32)
- *Modify a point defining a four point volume* (on page 33)
- *Modify the face of a volume* (on page 39)

Place a volume by four points

1. Click **Place Volume by Four Points** on the vertical toolbar.

   *The **Place Volume by Two Points** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

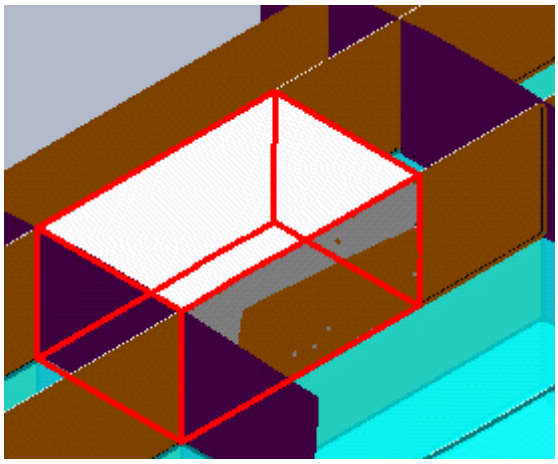
4. Select a folder from the **Space folder** list.

   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Select three points to define a plane for the volume. Then, select a fourth point to define the elevation from your defined plane for the volume.

The software creates the volume.



1. If necessary, click **Select Volumes for Addition** or **Select Volumes for Subtraction** to add to or subtract from the resultant volume.

NOTES

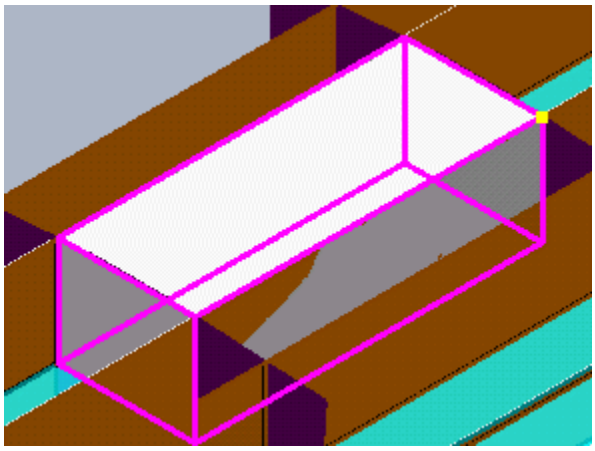
- You can also define the rectangular volume by using **PinPoint** to select coordinates for the points. For more information, see *PinPoint* and *Place objects using rectangular coordinates* in the *Common User's Guide*.
- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*
- By default, the global coordinate system is used to orient the rectangular volume relative to the placement points; however, you can use the **Coordinate System** list on the **PinPoint** ribbon prior to placement to select a different active coordinate system. A placed volume object maintains a local coordinate system that is used in future edits of the volume object. The orientation of the volume object does not change if the active coordinate system changes.

Modify a point defining a four point volume

1. On the vertical toolbar, click **Select**.
2. Select the volume to modify.

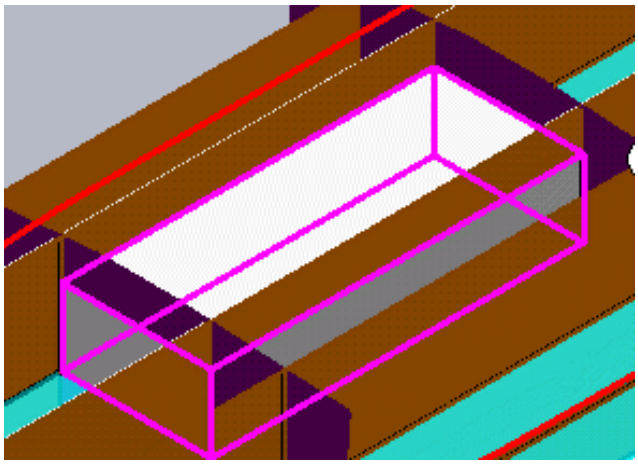
   *The volume highlights, and the **Modify** ribbon displays.*
3. Click **Volume Point 1**, **Volume Point 2**, **Volume Point 3**, or **Volume Point 4**.

The point displays in the graphic view.



4. Select a new location for the point.

The software modifies the volume.



Place Volume by Window

Defines a volume based on the active view boundaries of the model. You can use this command to quickly create a volume of the active view. The active view is defined by one of the following methods:

- By clipping - The software ignores the orientation of the clipped boundaries in the graphic view. A typical example is clipping by objects, and then placing a volume at the boundaries of the clipped area.
- Without clipping - The volume is based on the boundaries of the graphic view window, and projected out to limits of objects in the view.

You can edit the resulting volume in the same way as a volume by two points.

Place Volume by Window Ribbon

Sets options for placing a volume defined by an existing graphic view of the model.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Finish

Places the volume.

Name

Displays the default name for the volume object (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space Folder

Allows you to assign the new volume to a folder in the space hierarchy. Selecting **More** from the list opens the *Select Space Folder Dialog Box* (on page 17) from which you can select a folder.

Modify Volume by Two Points Ribbon

Sets options for modify volume objects defined by opposite vertices using the **Place Volume by Two Points** command.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Volume Point 1

Defines the first point for the definition of the volume.

Volume Point 2

Defines the second point for the definition of the volume.

Handles

Allows you to drag the sides of the volume to resize it.

Select Volumes for Addition

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

Select Volumes for Subtraction

Allows you to select volumes (graphically or from the **Workspace Explorer**), and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Finish

Places the volume as specified.

Name

Displays the default name for the volume object (as dictated by the active name rule), and allows you to type a different name, if needed. Names must be unique across the entire model.

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) dialog box from which you can select a type.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

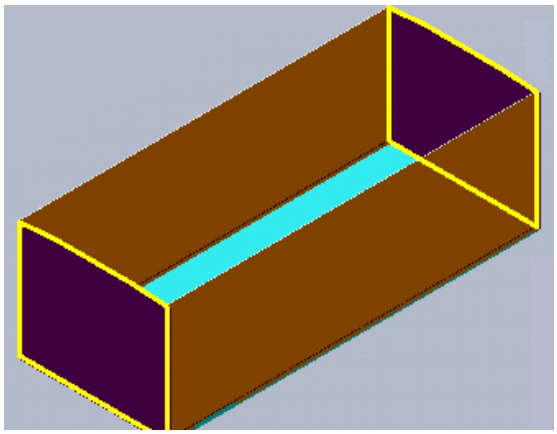
What do you want to do?

- *Place a volume by a clipped view* (on page 36)
- *Place a volume by a window view* (on page 37)
- *Modify a point defining a two point volume* (on page 27)
- *Modify the face of a volume* (on page 39)

Place a volume by a clipped view

1. Select a model object to include in the clipped view, such as a plate, profile, or equipment. Press CTRL and select additional model objects to include in the clipped view, then click **Clip by Object** on the **Common** ribbon.

   The active view boundaries clip to the selected objects.



1. Click **Place Volume by Window** on the vertical toolbar.

   *The **Place Volume by Window** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

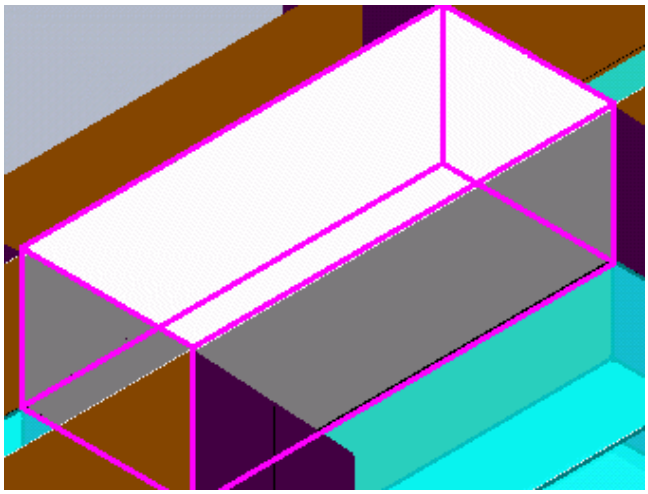
   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Click **Finish**.

   The software creates the volume.

   TIP Click **Clear Clipping** to better see the volume.



NOTE The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

Place a volume by a window view

1. Click **Place Volume by Window** on the vertical toolbar.

   *The **Place Volume by Window** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

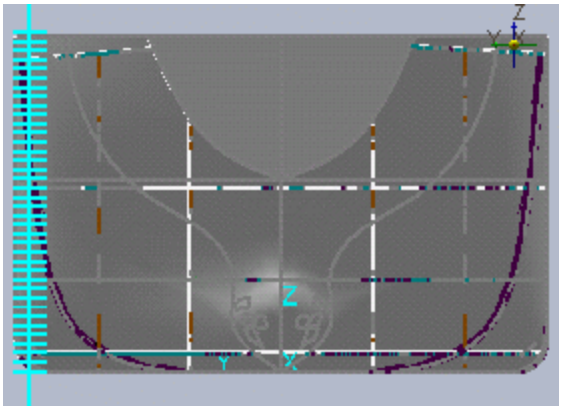
TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

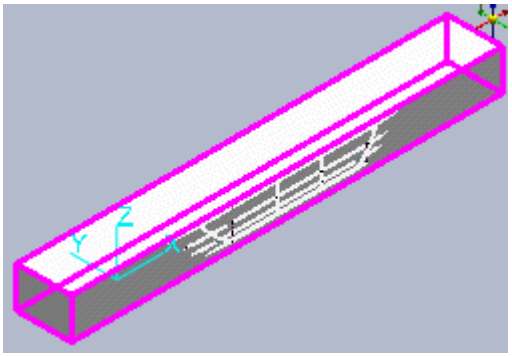
5. On the **Common** ribbon, use view controls such as **Common Views** to orient the view. Typically, you want an orthogonal plan, section, or elevation view. Then, click **Fit** on the **Common** ribbon to fit the view to the window.



6. Click **Finish.**

   The software creates the volume by performing a normal projection of the view.

   TIP Use the view controls to better see the volume.

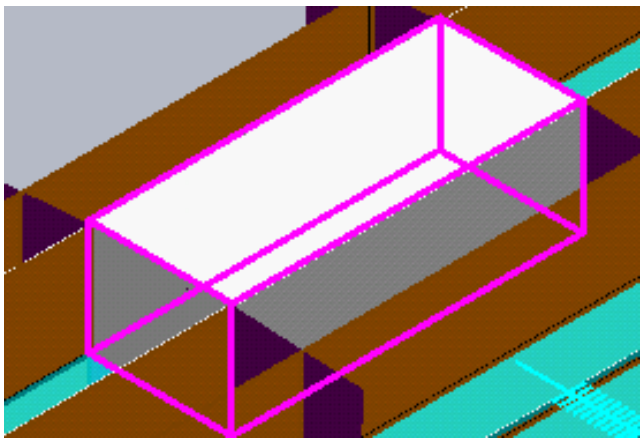


NOTE The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide*, available from **Help > Printable Guides**.

Modify the face of a volume

1. On the vertical toolbar, click **Select** .
2. Select the volume to modify.

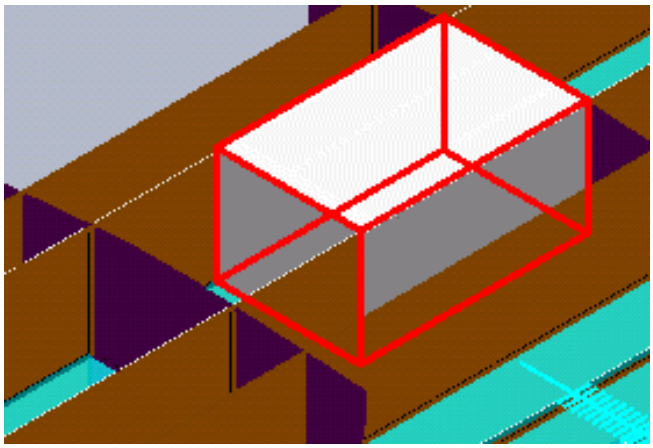
   *The volume highlights, and the **Modify** ribbon displays.*



3. Click **Handles** .
4. Select the face of the volume to modify, and drag the side to the new location.

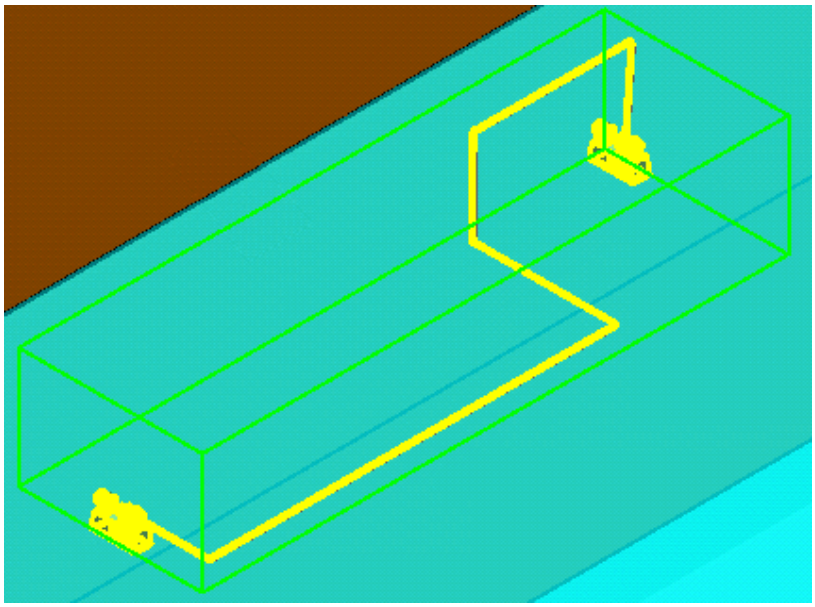
   TIP Use **QuickPick** to assist in selection of the required face.

   The software modifies the volume.



Place Volume by Selectset

Defines a compartment by placing a rectangular volume around a set of selected objects, such as structure, routing, and equipment objects. You can edit the resulting volume in the same way as a volume by two points.



Place Volume by Selectset Ribbon

Sets options for placing a volume defined by the range around a set of objects in the model.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Select object(s)

Allows you to pick objects in a graphic view or in the **Workspace Explorer**.

Finish

Places the volume.

Name

Displays the default name for the volume object (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space Folder

Allows you to assign the new volume to a folder in the space hierarchy. Selecting **More** from

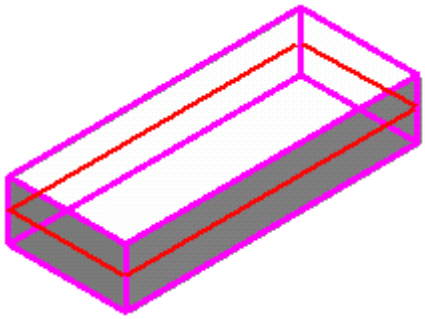
the list opens the *Select Space Folder Dialog Box* (on page 17) from which you can select a folder.

Volume Height

Allows you to type the length or the depth of the volume you want to create when you select coplanar linear objects such as grid lines, edges, and connections.

When you select one linear object, volume height is the length and width of the volume. The line is at the centerline of the volume and the length of the line is the length of the volume.

When you select two or more linear objects, volume height is the depth of the resulting volume. The depth is centered about the plane of the linear objects.



Modify Volume by Two Points Ribbon

Sets options for modify volume objects defined by opposite vertices using the **Place Volume by Two Points** command.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Volume Point 1

Defines the first point for the definition of the volume.

Volume Point 2

Defines the second point for the definition of the volume.

Handles

Allows you to drag the sides of the volume to resize it.

Select Volumes for Addition

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the

Workspace Explorer, and then click **Add** to add the volume to the list.

Select Volumes for Subtraction

Allows you to select volumes (graphically or from the **Workspace Explorer**), and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Finish

Places the volume as specified.

Name

Displays the default name for the volume object (as dictated by the active name rule), and allows you to type a different name, if needed. Names must be unique across the entire model.

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) dialog box from which you can select a type.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

What do you want to do?

- *Place a volume by selecting model objects* (on page 42)
- *Place a volume by defining volume height* (on page 44)
- *Modify a point defining a two point volume* (on page 27)
- *Modify the face of a volume* (on page 39)

Place a volume by selecting model objects

1. Click **Place Volume by Selectset** on the vertical toolbar.

   *The **Place Volume by Selectset** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

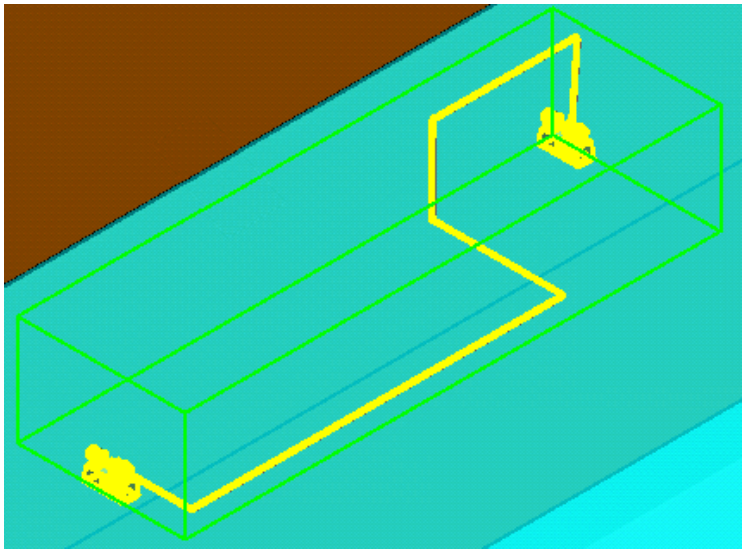
   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Select objects in the model.

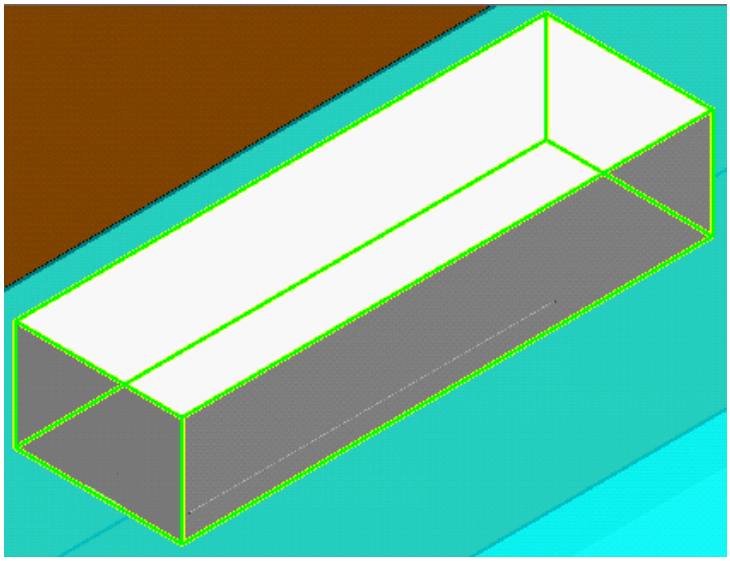
   TIP You can select the objects using a filter if necessary.

   A preview of the volume displays, changing as you select each new object.



6. Click **Finish**.

   The software creates the volume.



NOTES

- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*
- By default, the global coordinate system is used to orient the rectangular volume relative to the placement points; however, you can use the **Coordinate System** list on the **PinPoint** ribbon prior to placement to select a different active coordinate system. A placed volume object maintains a local coordinate system that is used in future edits of the volume object. The orientation of the volume object does not change if the active coordinate system changes.

Place a volume by defining volume height

1. Click **Place Volume by Selectset** on the vertical toolbar.

   *The **Place Volume by Selectset** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

   TIPS

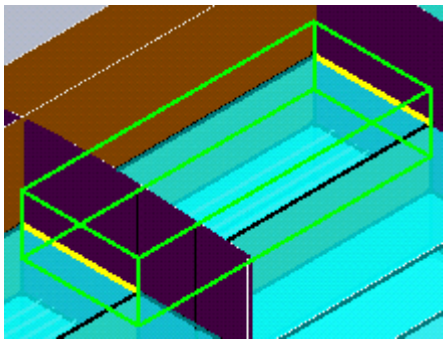
   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Select one or more linear objects in the model, such as grid lines, edges, or connections. Multiple linear objects must also be coplanar.

   TIP You can select the objects using a filter if necessary.

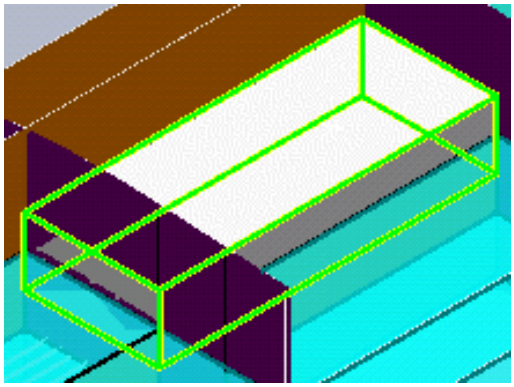
6. Type a value in the **Volume Height** box.

   A preview of the volume displays.

7. Click **Finish**.

The software creates the volume.



NOTES

- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*
- By default, the global coordinate system is used to orient the rectangular volume relative to the placement points; however, you can use the **Coordinate System** list on the **PinPoint** ribbon prior to placement to select a different active coordinate system. A placed volume object maintains a local coordinate system that is used in future edits of the volume object. The orientation of the volume object does not change if the active coordinate system changes.

Create Imported Volume

Imports an ACIS or an IGES file into the model in order to create volumes. The ACIS or the IGES file is typically created using third-party applications such as Napa, TID, or Proteus. For more information about ACIS and IGES files, see the *Molded Forms User's Guide*, available from the **Help > Printable Guides** command in the software.

NOTE This functionality is similar to the **Import Plate System** command in the Molded Forms task.

While using this command, you can associate the volume with a design specification from the catalog database. You can also add or subtract volumes to or from the overall volume.

Create Imported Volume Ribbon

Sets options for creating imported volumes from an ACIS or an IGES file.

Properties

Displays the **Volume Properties** dialog box, which allows you to set properties for the volume that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Select the File

Select the .SAT or the .IGS file to import.

Move the Object

Places the volume in the model.

Select Volumes for Addition

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

Select Volumes for Subtraction

Allows you to select volumes (graphically or from the **Workspace Explorer**) and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** from the dropdown list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

What do you want to do?

- *Import a volume from an ACIS or IGES file* (on page 46)

Import a volume from an ACIS or IGES file

1. On the vertical toolbar, click **Create Imported Volume** .

   *The **Import Compartment** dialog box displays.*
2. Select the .SAT or the .IGS file to import.
3. Click **Open**.

   TIPS

   - If the .SAT or the .IGS file does not generate a closed volume, the software displays a message instructing you to choose a different file.
   - After you choose a valid file, a volume appears in dynamics in the graphic view. The volume is attached to the mouse pointer at the centroid of the volume.
4. Specify a name for the volume in the **Name** box.
5. Select a volume type from the **Type** list.

TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

6. Select a folder from the **Space folder** list.

   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

7. In a graphic view, select a point to place the volume.

   The software places the volume at the volume origin.

   TIPS

   - You can use SmartSketch relationship indicators such as intersection and point on surface to assist in positioning the origin of the volume. For more information on the available relationship indicators, see *SmartSketch Options* in the *Common User's Guide*.
   - You can also define the volume origin by using **PinPoint** to select coordinates for the point. For more information, see *PinPoint* and *Place objects using rectangular coordinates* in the *Common User's Guide*.

8. If necessary, click **Select Volumes for Addition** or **Select Volumes for Subtraction** to add to or subtract from the resultant volume.
9. Continue placing the same volume in additional locations as required.

NOTES

- You can open a different .SAT or the .IGS file, if necessary, by clicking **Select the File** on the ribbon.
- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- After importing a volume, you can move the volume using the **Move** command. For precision, use **PinPoint** when moving the volume. For more information about these commands, see the *Common User's Guide*, available from the **Help > Printable Guides** command in the software.

Place Volume Along Path

Sketches a path in three-dimensional space, defines a cross-section, and projects the cross-section along the path to define a volume. The path can consist of straight lines, arcs, or even breaks in a line.

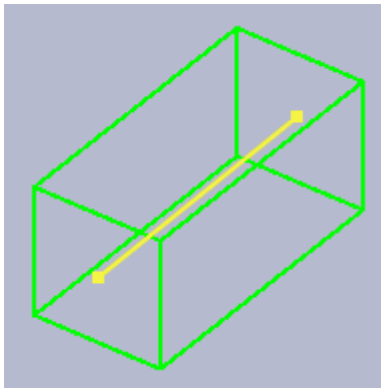
While using this command, you can associate the volume with a design specification from the catalog database. You can also add or subtract volumes to or from the overall volume.

This command is especially useful for volumes that need to have an unusual shape or that must negotiate around specific objects in the model. For example, if you need to leave adequate

space for equipment to reach a particular location such as fire evacuation route, **Place Volume Along Path** allows you to specify the path along which the equipment needs to travel as well as the appropriate amount of clearance that must be left on all sides.

When you place a volume along a path, a two-dimensional cross-section is projected along the path that you specify. The path determines the location of the volume in the model. The cross-section defines the shape and dimensions of the volume.

The following picture shows a path (yellow) and a rectangular cross-section projected along that path, with the resulting volume shown in green.



Defining Paths

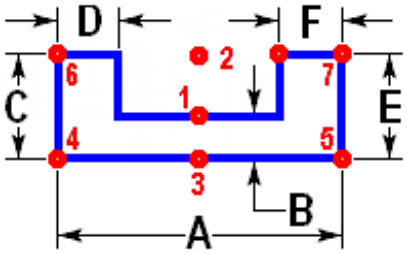
When you define the path along which a cross-section is projected, you can choose from a straight line or an arc. Or, if you want to break the path at a particular point, you can select **No Line** . You can also control characteristics of the path by specifying the types of turns (bend, cornice, or chamfer), the dimensions of the turns, and the plane for the path.

Defining Cross-Sections

When you define the cross-section for the volume, you can select from a standard set of cross-sections defined in the reference data, or you can sketch your own two-dimensional cross-section.

When you use a standard cross-section type, you can change the dimensions and the cardinal point on the **Cross-Section** tab on the **Volume Properties** dialog box. The cardinal point of the cross-section is important in determining the shape and location of the resulting volume. The cardinal point is the point at which you want the software to attach the cross-section to the path. All standard cross-section types have cardinal points from which you can choose.

For example, in this image, you can see that the default cardinal point **1** is located in the center of the cross-section. If you select another cardinal point, the resulting volume and the location of the volume along the path are different.



When you sketch a cross-section, you must sketch the cross-section on the two-dimensional plane that is orthogonal to the first leg of the path. The software displays this plane, which is perpendicular to the path, as you sketch the cross-section. The cardinal point also displays as you sketch.

Editing Customized Volumes

After creating customized volumes, you can edit them. The edit path ribbons allow you to modify the arcs, turns, or straight segments that are part of an existing path, either at placement or afterward.

Create Path Ribbon

Sets options for defining a new path.

Properties

Displays the **Sketch Properties** dialog box, which allows you to view properties for the path. For more information, see *Sketch Properties Dialog Box* (on page 153).

Finish Path

Displays the path in the active graphic view, and returns to the **Place Volume Along Path** command.

Cancel

Cancels the changes that you have made, and returns you to the model.

Edit

Allows you to modify and move the existing path. When you initially create a path, this option is only available after you place at least two points in the path. You can select a segment, a turn, or multiple segments to which you want to make modifications.

Create

Sketches the path or adds segments to an existing path.

Origin

This option is not used.

Reference Point

Specifies that you are defining the first point of the straight or arc path segment.

Second Point (Arc)

Specifies that you are defining the second point of an arc. This option only displays when you select **Arc by Three Points** or **Arc by End Points** in the **Path Type** list.

End Point

Specifies that you are defining the last point of a straight or arc path segment.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, select a different type in the **Path Type** list.

- **Line** - Specifies a straight line for the segment.

- **Arc by 3 Points** - Specifies an arc by three points for the segment. The first point is the start of the arc, the second point is a point on the arc, and the third point is the end of the arc.
- **No Line** - Specifies that you do not want the current segment of the path to have a line associated with it.
- **Arc by End Points** - Specifies an arc by end points for the segment. The first point you click is the start of the arc; the second point you click is the end of the arc; the third point you click defines the plane, radius, and sweep angle of the arc.
  - **Elliptical Arc** - Specifies an elliptical arc for the segment. The first point you click is the start of the arc; the second point you click is the center of the ellipse; the third point you click defines the minor/major axis ratio and the sweep angle.

Plane

Activates options for selecting a working plane for the path.

- **Plan Plane** - Defines the work surface as the XY plane. You can also press CTRL+1 to select this option.
- **Z Plane: X-West / Elevation Plane: East-West** - Defines the work surface as the XZ plane. You can also press CTRL+2 to select this option.
- **Section Plane: Y- South / Elevation Plane: North-South** - Defines the work surface as the YZ plane. You can also press CTRL+3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You can also press CTRL+4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points you define. You can also press CTRL+5 to select this option.
  - **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You can also press CTRL+6 to select this option.

Lock Angle

Locks or unlocks the **Angle** box. Locking the corresponding angle value creates a constraint along which the selected turn angle can be moved.

Angle

Specifies the angle for the turn.

Lock Length

Locks or unlocks the **Length** box.

Length

Specifies the length of the selected path segment.

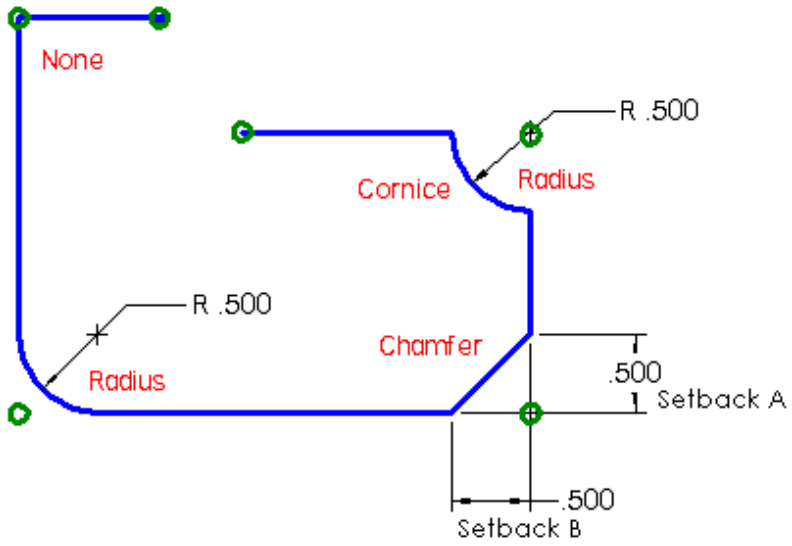
Turn Type

Specifies the type of turn associated with the current path segment. The **Turn Type** option is not available if you select **Arc** or **No Line** in the **Line Type** list.

- **None** - Indicates that no special turn type is applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.

- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback A and setback B of the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.

The following graphic includes an example of each of the available turn types:



Value

Defines dimensions for the selected turn type.

Offset

Type or select the offset between the path and the working plane.

Lock Radius

Locks or unlocks the **Radius** box. This option appears for the **Arc by End Points** path type.

Radius

Specifies the distance between the center of the arc and an end point of the arc. This option appears for the **Arc by End Points** path type.

Lock Minor/Major

Locks or unlocks the **Minor/Major** box. This option appears for the **Elliptical Arc** path type.

Minor/Major

Shows the ratio between the minor and major axes for the arc. This option appears for the **Elliptical Arc** path type.

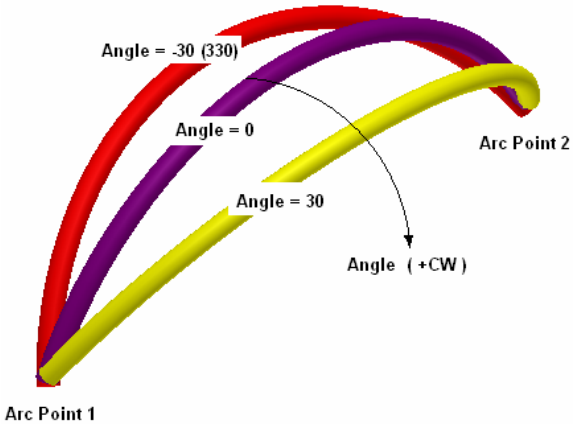
Lock Sweep

Locks or unlocks the **Sweep** box. This option appears for the **Arc by End Points** path type.

Sweep

Specifies the clockwise angle of the arc as looking from endpoint 1 to endpoint 2 of the arc.

This option appears for the **Arc by End Points** and **Elliptical Arc** path types.



Place Volume Along Path Ribbon

Sets options for placing volumes defined by a path and a cross section projected along the path.

Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Path

Displays the Create Path Ribbon, which allows you to define the path along which the volume will be created.

Cross-Section

Sets properties for the cross section. If one of the standard cross-section types is selected in the **Cross-Section** box, this button displays the *Cross-Section Tab* (on page 137) of the **Volume Properties** dialog box to allow you to modify the properties of the cross section. If you select **Sketch** as the cross-section type, this button displays the **Create Path** ribbon to allow you to sketch the two-dimensional cross section. The **Cross-Section** button is only available after you define a path for the volume.

Select Volumes for Addition

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

Select Volumes for Subtraction

Allows you to select volumes (graphically or from the **Workspace Explorer**) and then remove the selected volume from the volume definition.

Volume List

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Finish

Places the volume along the path specified with the cross section.

Cross-Section

Specifies the type of cross section that you want to project along the specified path. You can either select a standard cross-section defined in the reference data, or you can sketch your own custom cross-section.

Name

Displays the default name for the space (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the reference data used for determining the properties of the volume.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Edit Path Ribbon

Sets options for modifying a straight segment of an existing path. This ribbon appears when you select a straight segment in the existing path.

 **Properties**

Displays the **Sketch Properties** dialog box, which allows you to view properties for the path. For more information, see *Sketch Properties Dialog Box* (on page 153).

Finish Path

Displays the path in the active graphic view and returns you to the **Place Volume Along Path** command.

Cancel

Cancels the changes that you have made and returns you to the model.

 **Show Cross Section View**

Opens a new window that is oriented to display the sketch (cross section) plane as viewed from the end of the first path segment. The option is only available when the cross-section type is defined as **Sketch**.

Edit

Allows you to modify and move the existing path. You can select a segment, a turn, or multiple segments to which you want to make modifications.

Create

Displays the **Create Path** ribbon to allow you to add segments to the ends of an existing path. When editing a **Sketch** cross-section, you must first delete a segment before using **Create** to add segments because a **Sketch** cross-section has a closed path.

Origin

This option is not used.

Reference Point

Specifies that you are defining the first point of the straight or arc path segment.

Second Point (Arc)

Specifies that you are defining the second point of an arc. This option only displays when you select **Arc by Three Points** or **Arc by End Points** in the **Path Type** list.

End Point

Specifies that you are defining the last point of a straight or arc path segment.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list.

- **Line** - Specifies a straight line for the segment.
- **Arc by 3 Points** - Specifies an arc by three points for the segment. The first point is the start of the arc, the second point is a point on the arc, and the third point is the end of the arc.
- **No Line** - Specifies that you do not want the current segment of the path to have a line associated with it.
- **Arc by End Points** - Specifies an arc by end points for the segment. The first point you click is the start of the arc; the second point you click is the end of the arc; the third point you click defines the plane, radius, and sweep angle of the arc.
  - **Elliptical Arc** - Specifies an elliptical arc for the segment. The first point you click is the start of the arc; the second point you click is the center of the ellipse; the third point you click defines the minor/major axis ratio and the sweep angle.

Plane

Activates options for selecting a working plane for the path.

- **Plan Plane** - Defines the work surface as the XY plane. You can also press CTRL+1 to select this option.
- **Z Plane: X-West / Elevation Plane: East-West** - Defines the work surface as the XZ plane. You can also press CTRL+2 to select this option.
- **Section Plane: Y- South / Elevation Plane: North-South** - Defines the work surface as the YZ plane. You can also press CTRL+3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You can also press CTRL+4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points you define. You can also press CTRL+5 to select this option.
  - **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You can also press CTRL+6 to select this option.

Delete Selected Items

Deletes the selected path segments.

/ Length Locked

Defines whether or not the length of the selected linear segment should remain constant while moving.

- When locked , the software automatically modifies the turn points, along with the length and angle of the adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
- When not locked , the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

Angle 1

Displays the first angle used in the selected turn, if one exists. This option is read-only.

Angle 2

Displays the second angle used in the selected turn, if one exists. This option is read-only.

Angle 3

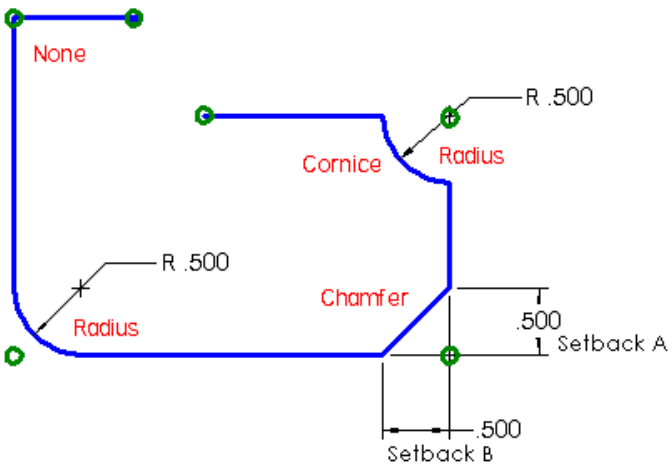
Displays the third angle used in the selected turn, if one exists. This option is read-only.

Turn Type

Specifies the type of selected turn. You can change the turn type by selecting another type in the list.

- **None** - Indicates that no special turn type is applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.
- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback A and setback B of the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.

The following graphic includes an example of each of the available turn types:



Value

Defines dimensions for the selected turn type.

Lock Radius

Locks or unlocks the **Radius** box. This option appears for the **Arc by End Points** path type.

Radius

Specifies the distance between the center of the arc and an end point of the arc. This option appears for the **Arc by End Points** path type.

Lock Minor/Major

Locks or unlocks the **Minor/Major** box. This option appears for the **Elliptical Arc** path type.

Minor/Major

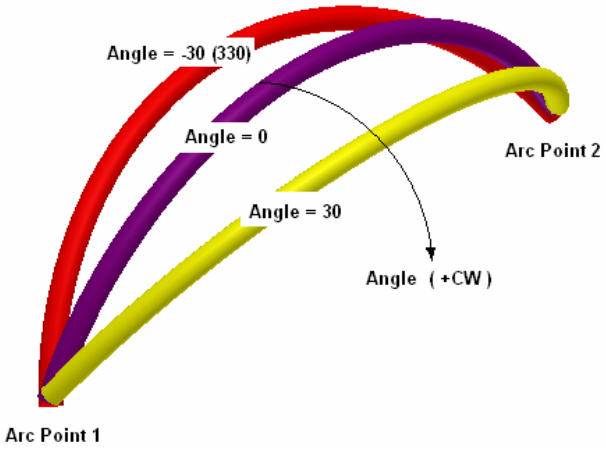
Shows the ratio between the minor and major axes for the arc. This option appears for the **Elliptical Arc** path type.

Lock Sweep

Locks or unlocks the **Sweep** box. This option appears for the **Arc by End Points** path type.

Sweep

Specifies the clockwise angle of the arc as looking from endpoint 1 to endpoint 2 of the arc. This option appears for the **Arc by End Points** and **Elliptical Arc** path types.



What do you want to do?

- *Place a volume along a path* (on page 57)
- *Define the path for a volume* (on page 58)
- *Define cross-section properties for a volume* (on page 60)
- *Sketch the cross-section for a volume* (on page 61)
- *Define a line* (on page 62)
- *Define an arc by three points* (on page 63)

- *Define an arc by end points* (on page 63)
- *Define an elliptical arc* (on page 64)
- *Move segments of a path* (on page 64)
- *Add segments to a path* (on page 64)
- *Modify a straight segment in a path* (on page 65)
- *Modify an arc in a path* (on page 66)
- *Modify a turn in a path* (on page 67)
- *Modify a sketched cross-section* (on page 67)
- *Convert a standard cross-section to a sketch* (on page 68)

Place a volume along a path

1. Click **Place Volume Along Path** on the toolbar.

   *The **Create Path** ribbon displays.*

2. Define points for the path of the volume.

   Define the path for a volume (on page 58)

3. On the **Place Volume Along Path** ribbon, select the cross-section type from the **Cross-Section** list.
4. Specify a name for the volume in the **Name** box.
5. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

6. Select a folder from the **Space folder** list.

   TIPS

   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16)..

7. In the **Cross-Section** box, select a cross-section type, and then click **Cross-Section** . Define cross-section properties by one of the following methods:

   - Select one of the standard cross-section types defined in the reference data.

     Define cross-section properties for a volume (on page 60)

     -OR-

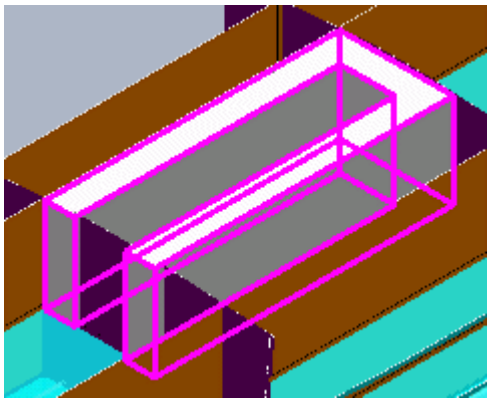
   - Select **Sketch** as the type and draw a two-dimensional cross-section.

     Sketch the cross-section for a volume (on page 61)

8. If necessary, click **Select Volumes for Addition** or **Select Volumes for Subtraction** on the ribbon to add to or subtract from the resultant volume.

9. Click **Finish**.

   The software creates the volume.



NOTES

- You can click **Volume Properties** on the ribbon to modify the volume properties.
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

Define the path for a volume

1. Click the first point for the path.
2. Select a segment type to create.

   Define a line (on page 62)

   Define an arc by three points (on page 63)

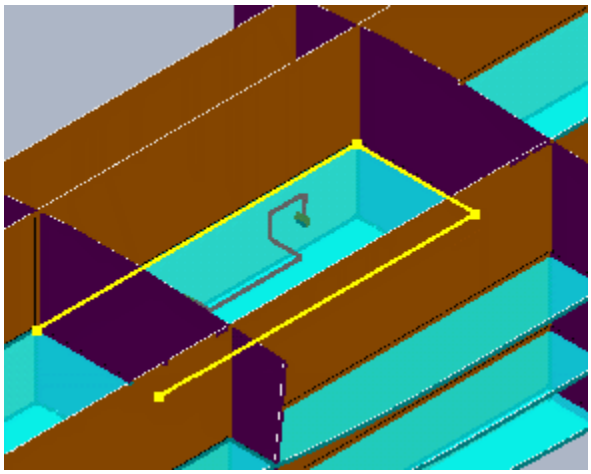
   Define an arc by end points (on page 63)

   Define an elliptical arc (on page 64)

   TIPS

   - You can use **PinPoint**, **Point Along**, and the SmartSketch relationship indicators (such as intersection and point on surface) when defining a path.
   - You can change the plane for the path at any time by selecting a different plane in the **Plane** list.
   - To break the path, click **No Line** in the **Path Type** list.
3. Place other segments of the path as required.

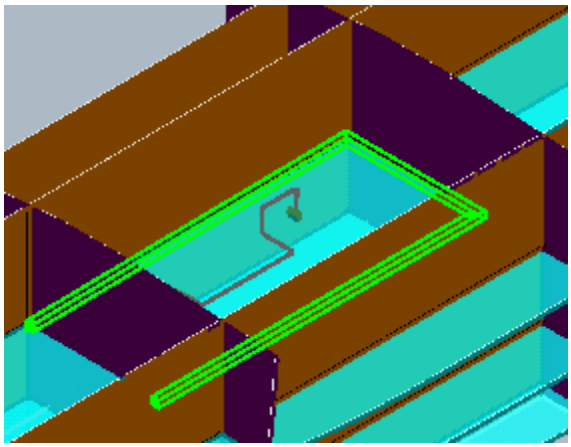
Path segments display as they are created.



TIP To change the turn type for a corner, click a different turn type in the **Turn Type** list, and define the angle or dimensions for the turn in the **Value** box. For more information, see *Define a line* (on page 62).

4. After you place all points that define the path, click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays. A preview of the volume, using the default or the last used cross-section, also displays.*



NOTES

- The software does not require that you close the path for a volume.
- After you place a segment of the path by defining two points, you can click **Edit** on the ribbon to change the segment.
- To manually set the angle and length for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.
- In the **Offset** box, you can specify the distance from the path segment to the working plane, if needed.

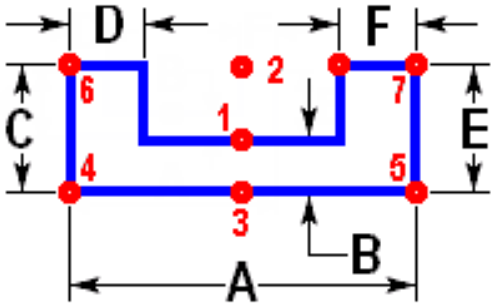
Define cross-section properties for a volume

1. In the **Cross-Section** box of the **Place Volume Along Path** ribbon, select a cross-section type, and then click **Cross-Section** ▭.

   *The **Compartment Volume Properties** dialog box displays.*

2. On the **Cross-Section** tab, define values for each dimension of the cross-section.

   TIP Click **Display Cross-Section Image** to view a graphic of the selected cross-section type. Cross-section images show the dimensions that you can define, the default cardinal points, and the angle for the cross-section. The following image shows the dimensions and cardinal points for a **Road** cross-section.

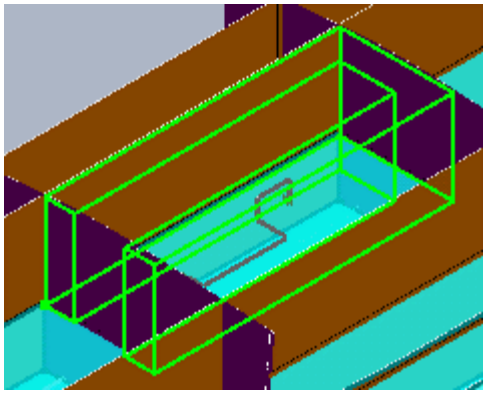


3. Select the cardinal point for the cross-section in the **Cardinality** list. The cardinal point is the point at which you want the software to attach the cross-section to the path.

   TIP When you change the cardinal point, the software automatically updates the volume display in the model.

4. Type an angle for the cross-section in the **Rotation Angle** box, if needed.
5. Click **Apply**.

   A preview of the volume displays showing the new property values.



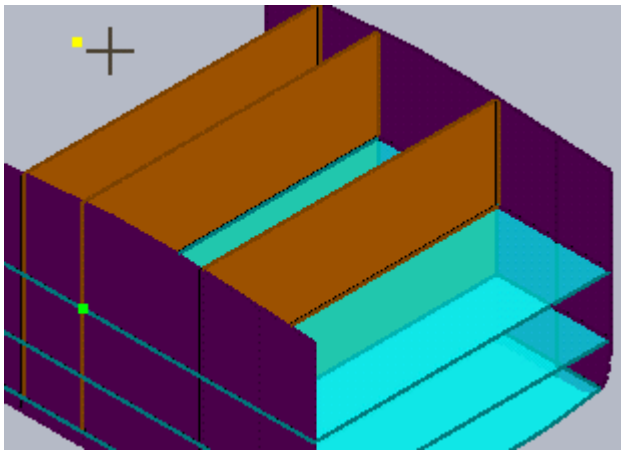
6. Click **OK**.

Sketch the cross-section for a volume

1. On the **Place Volume Along Path** ribbon, select **Sketch** in the **Cross-Section** box.
2. On the ribbon, click **Cross-Section** .

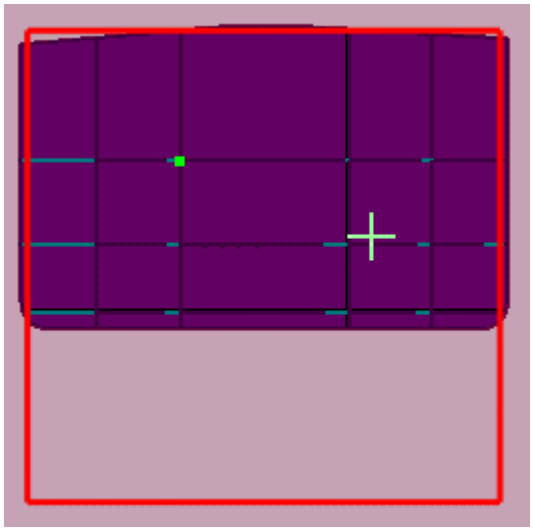
   *The **Edit Path** ribbon displays. The start point of the volume path displays in green and a yellow point displays at the cursor.*

   NOTE *The start point is also the cardinal point of the sketched cross-section.*



3. To show the cross-section plane in 2D in the graphic view, click **Show Cross-Section View** .

   The view moves to display the 2d plane of the cross-section. The start/cardinal point continues to display.



4. Click to create points and segments that define the cross-section. Click the starting point of the cross-section to close the cross-section.

   Define a line (on page 62)

   Define an arc by three points (on page 63)

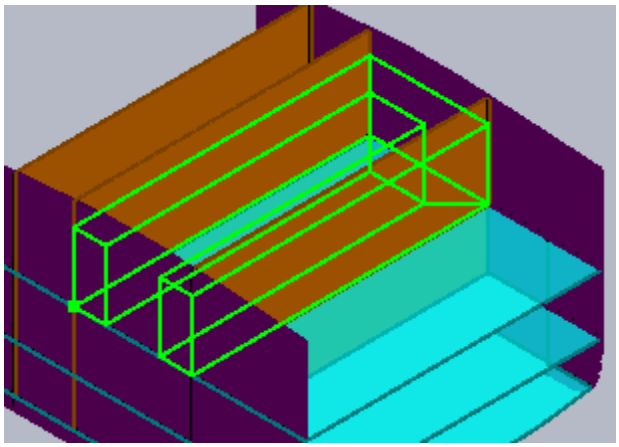
Define an arc by end points (on page 63)

Define an elliptical arc (on page 64)

5. Click **Finish Path**.

   TIP **Finish Path** is not available until you close the cross-section by clicking the starting point.

   A preview of the volume displays.



NOTES

- After you place a segment of the cross-section by defining two points, you can click **Edit** on the ribbon to change the segment.
- To manually set the angle and length for a segment of the cross-section, change the values in the **Angle** and **Length** boxes on the ribbon.
- The properties for a sketched cross-section include coordinates, turns, and related turn values, if applicable. You cannot modify the properties of a sketched cross-section on the **Compartment Volume Properties** dialog box. Instead, you must make changes to the path that defines the cross-section sketch. For more information, see *Modify a sketched cross-section* (on page 67).

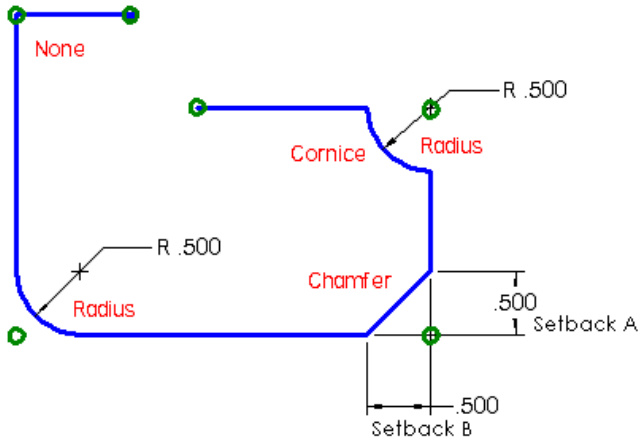
Define a line

1. In the **Path Type** list, click **Line** .
2. In the **Angle** box, type a value to constrain the angle of the line, if needed. You can also lock the angle.
3. In the **Length** box, type a value to constrain the length of the line, if needed. You can also lock the length.
4. In the **Turn Type** box, specify a turn, if needed. For example, you can select **Bend**, **Cornice**, or **Chamfer**.

   TIPS

   - You can specify a value for the dimensions of the turn in the **Value** box.

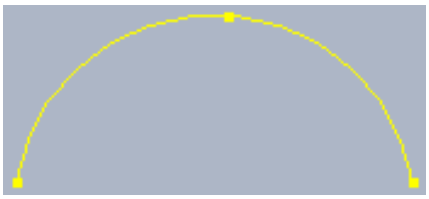
- The following graphic shows examples of the available turn types:



5. Click the end point for the line.

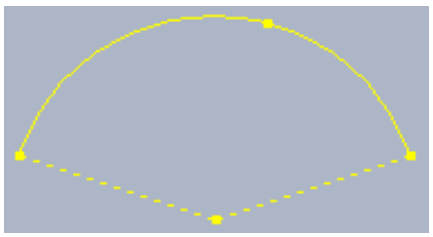
Define an arc by three points

1. In the **Path Type** list, click **Arc by 3 Points**.
2. Click to place the second point for the arc. This point lies on the arc.
3. Click to place the third and last point for the arc.



Define an arc by end points

1. In the **Path Type** list, click **Arc by End Points**.
2. Click to place the second end point for the arc.
3. Click to place a point on the arc.



NOTES

- You can use the **Radius** and **Sweep** boxes on the ribbon to control the parameters of the arc.
- The radius cannot be less than half the distance between the two end points of the arc.

Define an elliptical arc

1. In the **Path Type** list, click **Elliptical Arc** .
2. Click to place the center point of the ellipse.
3. Click to place the end point of the arc.

NOTES

- You can use the **Minor/Major** and **Sweep** boxes on the ribbon to control the parameters of the arc.
- The **Minor/Major** box shows the ratio between the minor and major axes of the ellipse.

Move segments of a path

1. On the vertical toolbar, click **Select** .
2. Select the volume for which you want to modify the path.

   TIP You can modify the path for volumes created with **Place Volume Along Path** , but not for volumes created with other volume commands.
3. On the ribbon, click **Path** .

   *The **Edit Path** ribbon displays.*
4. Select the segments that you want to move.
5. If you want the length of a straight segment to remain constant while you move the segment, click **Length Locked** .

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
6. Click to specify the starting location of the move vector.
7. Click to specify the ending location of the move vector.
8. Click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays.*
9. Click **Finish**.

Add segments to a path

1. On the vertical toolbar, click **Select** .
2. Select the volume for which you want to modify the path.

   TIP You can modify the path for volumes created with **Place Volume Along Path** , but not for volumes created with other volume commands.

3. On the ribbon, click **Path** .

   *The **Edit Path** ribbon displays.*

4. To add one or more segments to the path, click **Create**.
5. Click the point on the existing path at which you want to insert the new segment.

   TIPS

   - You can use **PinPoint** , **Point Along**, and the SmartSketch relationship indicators when defining your path.
   - You can change the plane for the path by selecting a different plane in the **Plane** list.
   - To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click three points to define the arc.
   - To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.
   - To break the path, click **No Line** in the **Path Type** list.

6. Click to place other points and add to the path as needed.

   TIP To change the turn type for the corner, click a different turn type in the **Turn Type** list, and define the dimensions for the turn in the **Value** box.

7. After you place all the points for the new segments, click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays.*

8. Click **Finish**.

NOTES

- The software does not require that you close the path for a volume. You can end the path wherever you want.
- You can click **Edit** on the ribbon to change the segment or modify the path further.
- To manually set the angle and length for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

Modify a straight segment in a path

1. On the vertical toolbar, click **Select** .
2. Select the volume for which you want to modify the path.

   TIP You can modify the path for volumes created with **Place Volume Along Path** , but not for volumes created with other volume commands.

3. On the ribbon, click **Path** .

   *The **Edit Path** ribbon displays.*

4. Select the straight segment that you want to modify.
5. To change the line type for the segment, click a new type in the **Path Type** list.
6. To change the plane for the segment, click a new plane in the **Plane** list.
7. To delete the segment, click **Delete** .

8. If you want the length of a straight segment to remain constant when you move the segment, click **Length Locked** .

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

9. Click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays.*

10. Click **Finish**.

NOTES

- You can also move the segments of a path. For more information, see *Move segments of a path* (on page 64).
- When the cross-sectional type is defined as **Sketch**, the **Show Cross-Section View** button is available on the edit ribbon.

Modify an arc in a path

1. On the vertical toolbar, click **Select** .
2. Select the volume for which you want to modify the path.

   TIP You can modify the path for volumes created with **Place Volume Along Path** , but not for volumes created with other volume commands.

3. On the ribbon, click **Path** .

   *The **Edit Path** ribbon displays.*

4. Select the arc that you want to modify.
5. To change the line type for the segment, click a different type in the **Path Type** list.
6. To change the plane for the segment, click a different plane in the **Plane** list.
7. To delete the segment, click **Delete** .
8. Click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays.*

9. Click **Finish**.

NOTES

- You can also move the segments of a path. For more information, see *Move segments of a path* (on page 64).

- When the cross-sectional type is defined as **Sketch**, the **Show Cross-Section View** button is available on the **Edit Path Arc** ribbon.

Modify a turn in a path

1. On the vertical toolbar, click **Select**.
2. Select the volume for which you want to modify the path.

   TIP You can modify the path for volumes created with **Place Volume Along Path**, but not for volumes created with other volume commands.
3. On the ribbon, click **Path**.

   *The **Edit Path** ribbon displays.*
4. Select the turn that you want to modify.
5. To change the plane for the turn, click a different plane in the **Plane** list.
6. To delete the turn, click **Delete**.
7. To change the turn type, select a different type in the **Turn Type** list.
8. To change the dimensions for the selected turn type, type a value in the **Value** box.
9. Click **Finish Path**.

   *The **Place Volume Along Path** ribbon displays.*
10. Click **Finish**.

NOTES

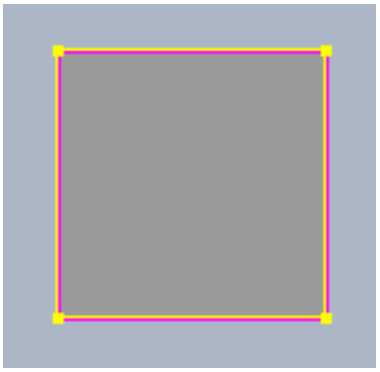
- You can also move the segments of a path. For more information, see *Move segments of a path* (on page 64).
- When the cross-sectional type is defined as **Sketch**, the **Show Cross-Section View** button is available on the **Edit Path Turn** ribbon.

Modify a sketched cross-section

1. On the vertical toolbar, click **Select**.
2. Select the volume placed along a path for which you want to modify the sketched cross-section.
3. On the **Place Volume Along Path** ribbon, click **Cross-Section**.

   *The **Edit Path** ribbon displays.*
4. To show the cross-section plane in 2D in the graphic view, click **Show Cross-Section View**.

The view moves to display the 2d plane of the cross-section.



5. Modify segments of the cross-section as required.

   Modify a straight segment in a path (on page 65)

   Modify an arc in a path (on page 66)

   Modify a turn in a path (on page 67)

   Move segments of a path (on page 64)

   Add segments to a path (on page 64)

   TIP You can add as many segments to the cross-section path as you need. However, you must close the path in order to save it.

6. If you want to delete a segment, select the segment that you want to delete, and then click **Delete** .

7. Click **Finish Path**.

   TIP The **Finish Path** button is not available until you close the cross-section by clicking the starting point.

   *The **Place Volume Along Path** ribbon displays.*

8. Click **Finish**.

NOTE To modify the properties of a sketched cross-section, click **Edit** > **Properties**, or click **Properties** on the ribbon. The properties for the cross-section include the coordinates, turns, and related turn values, if applicable. For more information, see *Sketch Properties Dialog Box* (on page 153).

Convert a standard cross-section to a sketch

1. On the vertical toolbar, click **Select** .
2. Select the volume for which you want to modify the cross-section.
3. On the ribbon, click **Cross-Section** to display the **Properties** dialog box.
4. On the **Cross-Section** tab, select **Sketch** in the **Cross-Section** list.
5. Click **OK**.

   The software converts the standard cross-section to a sketched cross-section. The shape of the original cross-section is broken down into the segments and points that define the path

of the cross-section.

6. Make modifications to the sketched cross-section as needed.

   Modify a sketched cross-section (on page 67)

NOTES

- When you convert a standard cross-section to a sketch, the properties for the sketched cross-section include the coordinates, the turn type, and values for each point that define the cross-sectional path. For more information, see *Sketch Properties Dialog Box* (on page 153).
- You cannot modify the properties of a sketched cross-section in the **Properties** dialog box. Instead, you must make changes to the path that defines the cross-sectional sketch.

Create Volume by Faces

 Creates a volume defined by faces, such as plate systems or reference planes. You can select the faces in three ways:

- One-by-one, or by fencing an area in the model.
- By 3D range box.
- By a point within the volume.

NOTE For complex volumes, you can use the **Face List**  to assist in specifying the faces.

Create Volume by Faces Ribbon

Sets options for placing a volume defined by faces and boundaries.

 **Volume Properties**

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

 **Select Faces**

Allows you to specify the faces for the volume definition.

Finish

Generates the defined volume and exits the command.

 **Select Faces**

Selects reference planes, grid planes, or plates. This method is the default.

 **Select Faces by 3D Range**

Allows you to specify a three-dimensional range box using two diagonal points. You can select faces inside and overlapping this range.

 Select Faces by Point

Allows you to specify a point. The surrounding faces of the smallest volume enclosing this point are selected. When there is more than one possible volume, this point is used to resolve the ambiguity.

Plate Filter

Restricts the selection of plates or grids to one of the following types:

- **Grid Planes and Plates** - All plate systems, grid planes, and reference planes.
- **All Plate Systems** - All plate systems.
- **Grid Planes** - All grid planes and reference planes.
- **Water Tight Plates** - Plate systems with the **Tightness** property defined as **Water Tight**.
- **Air Tight Plates** - Plate systems with the **Tightness** property defined as **Air Tight**.
- **Oil Tight Plates** - Plate systems with the **Tightness** property defined as **Oil Tight**.
- **Non Tight plates** - Plate systems with the **Tightness** property defined as **Non-Tight**.
- **More** - Opens the **Select Filter** dialog box, where you can define a custom filter. As an example, you can restrict selection to the reference planes of one coordinate system when multiple coordinate systems exist in the model. For more information, see *Select Filter Dialog Box* in the *Common User's Guide and Create a Filter for a Grid System* (on page 80).

 **Reject**

Removes the selected faces.

 **Accept**

Accepts the selected faces

 **Face List**

Displays the **Face List** dialog box, from which you can view information about the selected faces for the volume. You can use this dialog box to add or remove faces. For more information, see *Face List Dialog Box* (on page 74).

 **Select Volumes for Addition**

Allows you to select additional volumes (graphically or from the **Workspace Explorer**) to add to the current volume definition. First, select a volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add the volume to the list.

 **Select Volumes for Subtraction**

Allows you to select volumes (graphically or from the **Workspace Explorer**) and then remove the selected volume from the volume definition.

 **Volume List**

Displays the *Boolean Operation List Dialog Box* (on page 28), which lists the volumes added to or subtracted from the overall volume. You can add or remove volumes using the buttons on this dialog box.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** opens the *Select Volume Dialog Box* (on page 29), from which you can select a type.

Space Folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Modify Multiple Volumes by Faces Ribbon

Edits multiple by-face volumes that share a common boundary.

 **Volume Properties**

Displays the **Properties** dialog box, which allows you to set properties for the volumes that you are editing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

 **Select Faces**

Allows you to specify the faces for the volumes.

 **Reject Selected Volumes/Surfaces**

Removes the selected volumes or surfaces.

 **Accept Selected Volumes/Surfaces**

Accepts the selected volumes or surfaces.

Plate Filter

Restricts the selection of plates to a certain type, such as **Water Tight Plates**. Child plates and detail plates are not allowed. Select **More...** to specify a new filter.

Finish

Applies changes to the volumes and exits the command.

 **Face List**

Displays the **Faces List** dialog box, from which you can view information about the selected faces for the volumes. You can use this dialog box to add or remove faces. For more information, see *Face List Dialog Box* (on page 74).

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

What do you want to do?

- *Place a volume bound by faces* (on page 72)
- *Add faces using the Face List* (on page 74)

Place a volume bound by faces

1. On the vertical toolbar, click **Create Volume by Faces** .

   *The **Create Volume by Faces** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

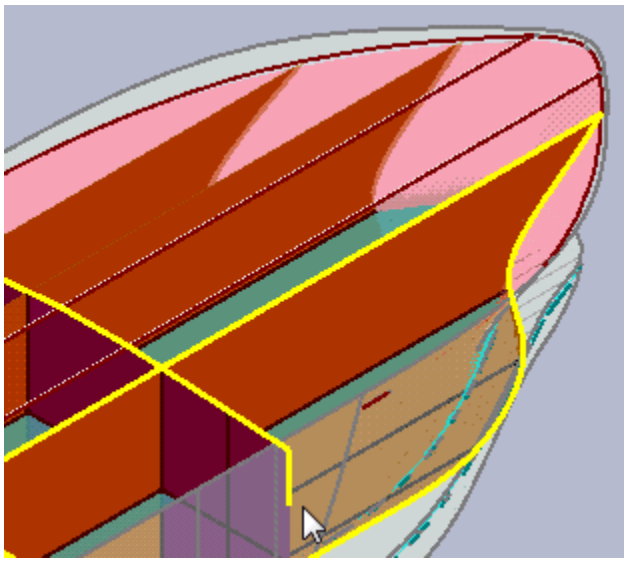
   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

4. Select a folder from the **Space folder** list.

   TIPS

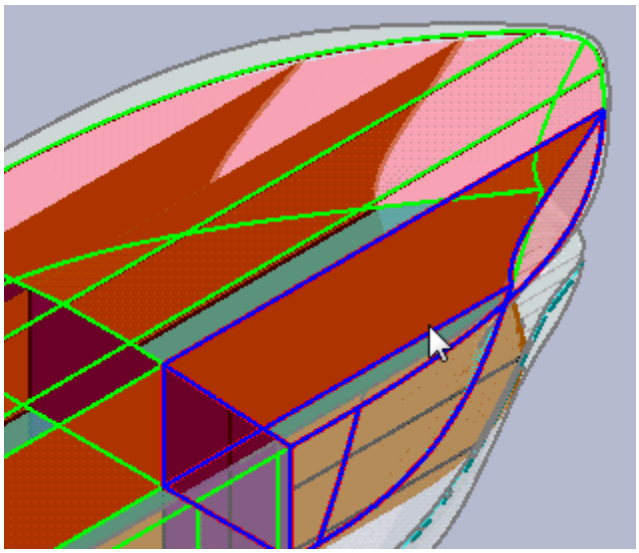
   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
   - Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Select the filter to use to restrict plate or grid selection in the **Plate Filter** box.
6. Select the faces for the volume, using **Select Faces** , **Select Faces by 3D Range** , or **Select Faces by Point** .



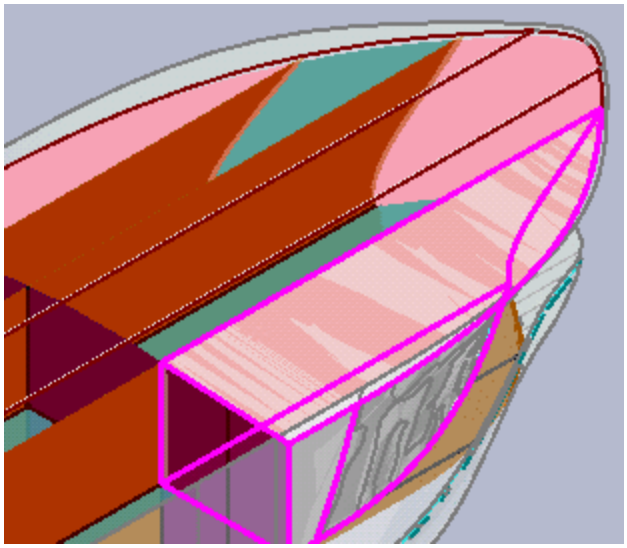
TIP You can select the faces in any order.

7. If there is more than one possible volume from the selected planes, select the required volume.



8. Click **Finish**.

   The software creates the volume.



NOTES

- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

- If you are creating a complex volume, you can use the **Faces List** to assist in specifying the faces and to define offsets from faces. For more information, see *Add faces using the Face List* (on page 74). You can also click **Accept** or **Reject** on the ribbon as you select faces.
- Optionally, click **Select Volumes for Addition** or **Select Volumes for Subtraction** on the ribbon to add to or subtract from the resultant volume.
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

Add faces using the Face List

The following procedure applies when you are creating volumes by faces or when creating multiple volumes.

1. On the ribbon, click **Face**.
2. Click faces such as bulkheads or decks in a graphic view or in the **Workspace Explorer**. The software adds the faces to the **Face List** dialog box.
3. Select a face on the dialog box, and click **Remove** if you change your mind about including that face.
4. Click **OK** on the **Face List** dialog box to return to the command.

Face List Dialog Box

Lists the selected surfaces and boundaries in a grid format when you are defining volumes. This dialog box is often helpful when you are constructing complex volumes.

Face Tab (Face List Dialog Box) (on page 74)

Bound Tab (Face List Dialog Box) (on page 74)

Face Tab (Face List Dialog Box)

Displays the selected faces for the volume. You can add faces to this list, or remove faces from the list.

Name

Displays the name of the face.

Offset

Type or select the offset between the selected plane and a bounding plane. The software creates the bounding plane at an offset from the selected plane when creating the volume.

Remove

Allows you to select faces (graphically or from the **Workspace Explorer**) and then remove the selected face from the bound volume definition.

Bound Tab (Face List Dialog Box)

Displays the bounding surfaces for a selected face. You select a face at the top of this tab, and then you can specify the boundaries for that face.

Faces

Select a face from this list, which is populated with the faces from the **Face** tab of this dialog box.

Name

Displays the name of the bounding face.

NOTE The order of the boundaries is important because the order determines how a face is trimmed.

SECTION 6

Create Multiple Volumes Simultaneously

The software provides two ways of creating multiple volumes at once.

With **Create Multiple Volumes**  , you specify a volume in the model, and then you split the volume by inner structure. The existing, parent volume is not deleted, but the volume is changed to a void space if the parent is the same type as the children volumes.

You can also create many volumes at once using **Tools** > **Create Compartments Automatically**. This command creates all the compartments in a ship by using all existing faces. You only define a filter for the faces.

When you are creating multiple volumes, it is often helpful to select a specific coordinate system. The **Plate Filter** box on the ribbon allows you to select a grid system for this purpose. For more information about plate filters, see *Create a filter for a grid system* (on page 80).

Create Multiple Volumes (on page 76)

Create Compartments Automatically (on page 81)

Create Multiple Volumes

 Creates multiple compartments based on a set of boundaries that make up a closed volume and a set of plates inside the closed volume that define individual compartments.

You specify the outer boundaries by faces, by 3D range, or by selecting an existing volume. The outer boundaries must form a closed geometry. You specify the inner structures by faces, by 3D range, or by all intersecting faces. The set of inner structures available for selection can be restricted with a plate filter.

NOTE This command is useful for creating many volumes in one part of the ship. To create all the volumes in the whole ship, use **Tools** > **Create Compartments Automatically**.

NOTE When you use **Create Multiple Volumes**  , the software changes the input volume to a void space if the input volume has the same type as the output volumes. The existing, parent volume is not deleted, but it is changed to a void space if the parent is the same type as the children volumes.

Create Multiple Volumes Ribbon

Sets options for defining a set of new volumes by specifying a set of boundaries that make up a closed volume and a set of plates inside the closed volume that define individual compartments.

 **Volume Properties**

Displays the **Properties** dialog box, which allows you to set properties for the volumes that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Select Outer Boundaries

Specifies the outer volume. The outer boundaries that you select must form a closed geometry. For example, you can select the hull and main deck.

- **Select Planes or Plates** - Allows you to specify faces to define the outer volume.
- **Select Faces by 3D Range** - Allows you to specify a three-dimensional range box using two diagonal points. You can select faces inside and overlapping this range.
- **Select Any Existing Volume** - Allows you to select an existing volume in the model.

Select Inner Boundaries

Allows you to select faces inside the outer volume.

- **Select Planes or Plates** - Allows you to specify faces to define the inner volumes.
- **Select Faces by 3D Range** - Allows you to specify a three-dimensional range box using two diagonal points. You can select faces inside and overlapping this range.
- **Select All Faces** - Allows you to specify the faces that intersect the outer volume chosen in **Select Outer Boundaries**.

Finish

Generates the defined volumes and exits the command.

Plate Filter

Restricts the selection of plates or grids to one of the following types:

- **Grid Planes and Plates** - All plate systems, grid planes, and reference planes.
- **All Plate Systems** - All plate systems.
- **Grid Planes** - All grid planes and reference planes.
- **Water Tight Plates** - Plate systems with the **Tightness** property defined as **Water Tight**.
- **Air Tight Plates** - Plate systems with the **Tightness** property defined as **Air Tight**.
- **Oil Tight Plates** - Plate systems with the **Tightness** property defined as **Oil Tight**.
- **Non Tight plates** - Plate systems with the **Tightness** property defined as **Non-Tight**.
- **More** - Opens the **Select Filter** dialog box, where you can define a custom filter. As an example, you can restrict selection to the reference planes of one coordinate system when multiple coordinate systems exist in the model. For more information, see *Select Filter Dialog Box* in the *Common User's Guide and Create a Filter for a Grid System* (on page 80).

Reject

Removes the selected boundaries or faces.

Accept

Accepts the selected boundaries or faces.

Face List

Displays the **Face List** dialog box, from which you can view information about the selected faces. You can use this dialog box to add or remove faces. For more information, see *Face List Dialog Box* (on page 74).

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space folder

Allows you to assign the new volumes to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Bound

Specifies if you want the volumes to behave as completely separate volumes.Select to modify the volumes as if they had been created individually. Clear to modify the resulting compartments as a group, in which the modify ribbon only allows you to make changes to the group as a whole.

What do you want to do?

- *Create multiple volumes in part of a ship* (on page 78)
- *Create a filter for a grid system* (on page 80)

Create multiple volumes in part of a ship

1. On the vertical toolbar, click **Create Multiple Volumes** .

   *The **Create Multiple Volumes** ribbon displays.*

2. Specify a name for the volume in the **Name** box.
3. Select a volume type from the **Type** list.

   TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

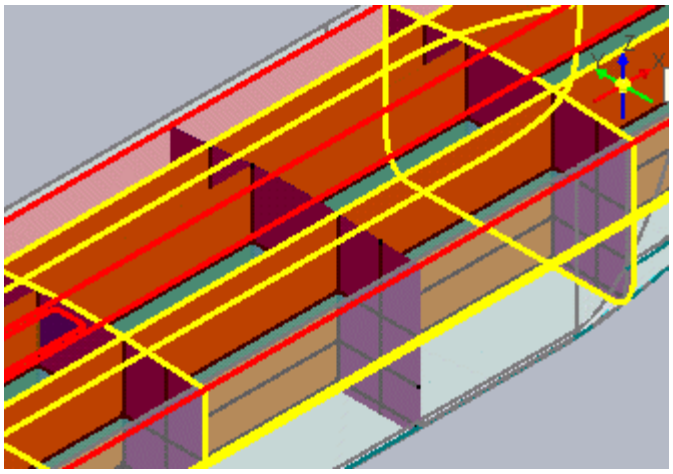
4. Select a folder from the **Space folder** list.

   TIPS

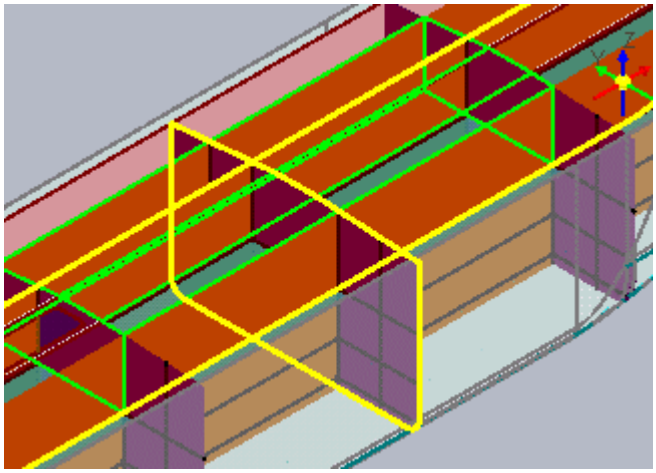
   - The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).

- Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

5. Select the filter to use to restrict plate or grid selection in the **Plate Filter** box.
6. Select the faces for the outer volume. The selected faces must define a closed boundary. Use **Select Planes or Plates**, **Select Faces by 3D Range**, or **Select Any Existing Volume**.

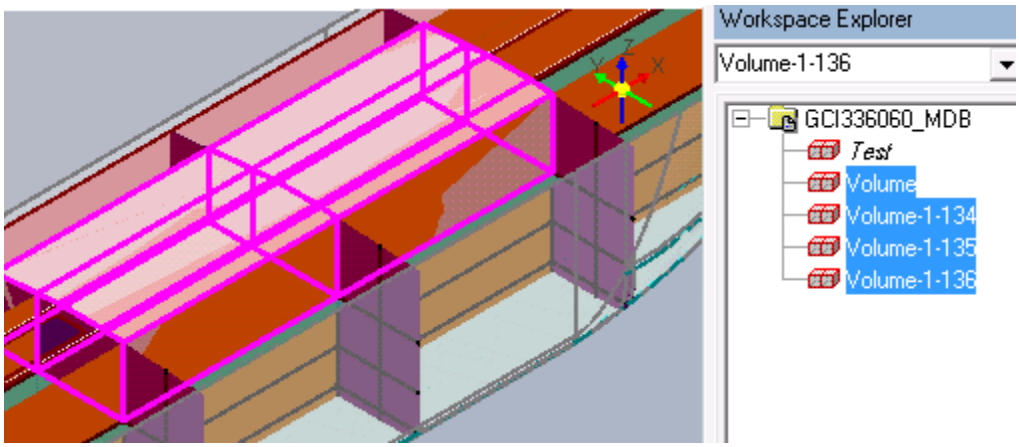


7. Click **Accept**, and then click **Select Inner Boundaries**.
8. If required, change the filter in the **Plate Filter** box.
9. Select the faces for the inner boundaries, using **Select Planes or Plates**, **Select Faces by 3D Range**, or **Select All Faces**.



10. Clear **Bound** if you want to modify the resulting volumes as a group.
11. Click **Finish**.

The software creates the volume.



NOTES

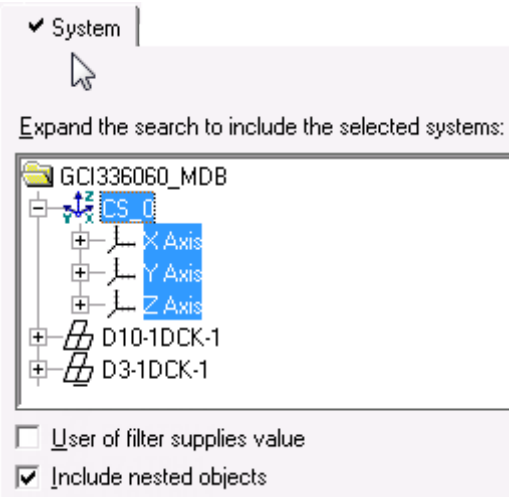
- If a plate contains an opening, the plate does not form a closed boundary.
- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- You can use the **Face List** to assist in specifying the outer boundaries and inner structure for the multiple volumes. For more information, see *Add faces using the Face List* (on page 74). You can also click **Accept** or **Reject** on the ribbon as you select faces.
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*
- The software changes the input volume to a void space if the input volume has the same type as the output volumes. The existing, parent volume is not deleted, but the volume is changed to a void space if the parent is the same type as the children volumes.

Create a filter for a grid system

This procedure describes how to create a filter to restrict selection to one grid coordinate system when multiple coordinate systems exist in the model. You can follow this procedure when using the Compartmentation commands **Create Multiple Volumes**, **Tools** > **Create Compartments Automatically**, and **Create Volume by Faces**.

1. Click **Tools** > **Select by Filter**.
2. On the **Select Filter** dialog box, click **New Filter**.

3. On the **New Filter Properties** dialog box, select a grid system on the **System**. Select **Include nested objects** and click **OK**.



   For more information about filters, see *Select Filter Dialog Box* in the *Common User's Guide*.

4. Start the command that you want (**Create Multiple Volumes**, **Tools** > **Create Compartments Automatically**, or **Create Volume by Faces**).
5. In the **Plate Filter** box on the ribbon, select **More**.
6. On the **Select Filter** dialog box, click the grid system filter that you just defined.

NOTE An alternative method is to redefine the workspace temporarily to only contain the one grid system needed (if more than one grid system exists in the model). Then, you can start the command you want, and select **Grid Planes** in the **Plate Filter** box on the ribbon.

Create Compartments Automatically

Tools > **Create Compartments Automatically** creates all the compartments in an entire ship in one operation using existing faces (plates or planes). You can specify a filter for the faces.

NOTE This command is useful for creating the volumes in an entire ship. To create volumes in just part of a ship, use **Create Multiple Volumes**.

Create Compartments Automatically Ribbon

Sets options for defining compartments within an entire ship.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the volumes that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Finish

Generates the defined volumes and exits the command.

Plate Filter

Restricts the selection of plates or grids to one of the following types:

- **Grid Planes and Plates** - All plate systems, grid planes, and reference planes.
- **All Plate Systems** - All plate systems.
- **Grid Planes** - All grid planes and reference planes.
- **Water Tight Plates** - Plate systems with the **Tightness** property defined as **Water Tight**.
- **Air Tight Plates** - Plate systems with the **Tightness** property defined as **Air Tight**.
- **Oil Tight Plates** - Plate systems with the **Tightness** property defined as **Oil Tight**.
- **Non Tight plates** - Plate systems with the **Tightness** property defined as **Non-Tight**.
- **More** - Opens the **Select Filter** dialog box, where you can define a custom filter. As an example, you can restrict selection to the reference planes of one coordinate system when multiple coordinate systems exist in the model. For more information, see *Select Filter Dialog Box* in the *Common User's Guide and Create a Filter for a Grid System* (on page 80).

Reject Selected Volumes/Surfaces

Removes the selected volumes or surfaces.

Accept Selected Volumes/Surfaces

Accepts the selected volumes or surfaces.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space folder

Allows you to assign the new volumes to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Create multiple volumes in an entire ship

1. Click **Tools** > **Create Compartments Automatically**.
2. In the **Plate Filter** box, specify a filter for the faces that separate the compartments. For more information about plate filters, see *Create a Filter for a Grid System* (on page 80).

   TIP You typically want to select plate filters, such **All Plate Systems** and **Water Tight Plates**. A filter that includes grids usually creates too many small compartments.
3. Specify a name for the volume in the **Name** box.
4. Select a volume type from the **Type** list.

TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

5. Select a folder from the **Space folder** list.

TIPS

- The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
- Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

6. Click **Finish**.

NOTES

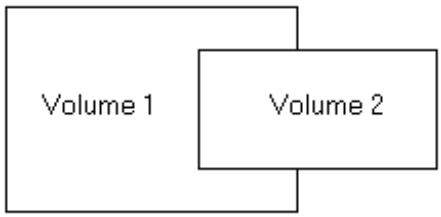
- You can click **Volume Properties**  on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

SECTION 7

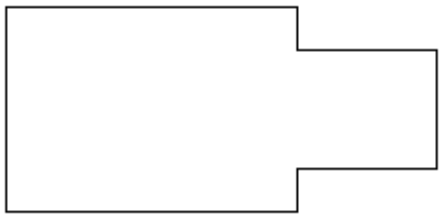
Create Volume Objects from Existing Volumes

The software provides two commands for creating a new volume object out of existing volumes: **Merge Volumes** and **Bound by Volumes**. Both commands allow you to select two or more existing volumes and join them to form a new volume.

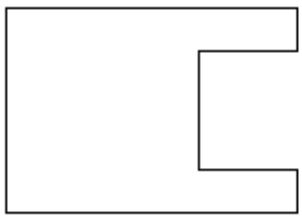
When you use these commands, you can add to or remove from the new volume. You can also specify the intersection between volumes. In the following pictures (plan view), two volumes are joined using different operations.



Sub-volumes: Volume 1 and Volume 2



Resultant volume with Add option



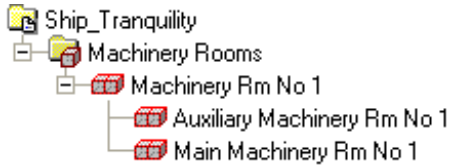
Resultant volume with Subtract option

Resultant volume with Intersect option

The main difference in these commands is in the way their sub-volumes are treated.

Creating Volumes from Merging Volumes

You can create a volume that is the union of two or more volumes using **Merge Volumes** . The existing volumes are consumed in the merge operation; consequently, the sub-volumes are no longer available as separate entities. The software removes them from the graphic views and groups them under the merged volume in the **Workspace Explorer**.

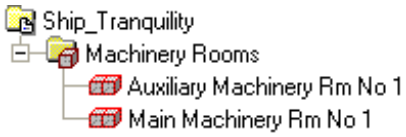


The resulting 3-D volume type is, by default, the same type as the last type used; however, you can change this type if necessary. The resulting volume inherits the properties of the first volume that you select for the merge.

You can only select the sub-volumes grouped under the merge from the **Workspace Explorer**. Sub-volumes used to create a merged volume have no value on their own and are used solely to create geometric shapes that otherwise could not be created. Deleting a merged volume deletes all of the sub-volumes associated with it.

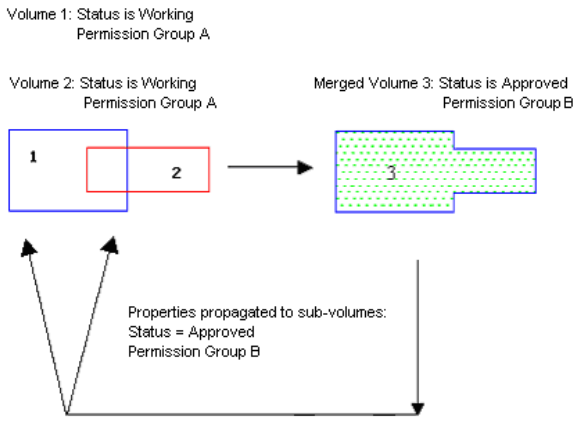
Creating Volumes Bound by Other Volumes

You can also create a volume from existing volumes using **Bound by Volumes** . The existing volumes remain graphically in the view and in the **Workspace Explorer**.



The sub-volumes that are selected to create the new volume can be selected graphically or from the **Workspace Explorer**. Deleting the new volume deletes the individual integrity of the sub-volumes but does not delete the sub- volumes.

After creating a volume using **Merge Volumes** or **Bound by Volumes** , you can modify the volume properties, such as location. If you modify the status or permission group of the resulting volume, then the change is propagated to the sub- volumes, as shown in the following illustration:



In this example, Volume 1 and Volume 2 have a status of Working and belong to permission group A. Their union volume is created with a status of Approved and assigned to permission group B. The approved status and permission group B are propagated back to Volumes 1 and 2.

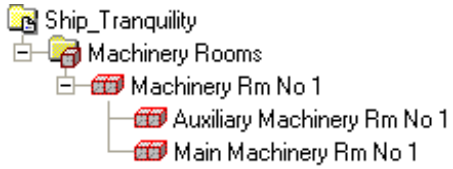
Merge Volumes (on page 86)

Bound by Volumes (on page 90)

Merge Volumes

Merges multiple existing volume objects into a single new volume. By using the **Operation** box to define the volume objects to add to or remove from the merged volume, you can place more complex volume shapes than with the standard placement commands. For example, with this command, you can create an L-shaped volume by merging two overlapping rectangular volume objects.

The existing volumes are consumed in the merge operation; consequently, the sub-volumes are no longer available as separate entities. The software removes them from the graphic views and groups them under the merged volume in the **Workspace Explorer**.



The resulting 3-D volume type is, by default, the same type as the last type used; however, you can change this type if necessary. The resulting volume inherits the properties of the first volume that you select for the merge.

You can only select the sub-volumes grouped under the merge from the **Workspace Explorer**. Sub-volumes used to create a merged volume have no value on their own and are used solely to create geometric shapes that otherwise could not be created. Deleting a merged volume deletes all of the sub-volumes associated with it.

Merge Volumes Ribbon

Sets options for merging existing volume objects into a single new volume.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. By default, the new merged volume inherits the properties of the first volume object that you select for the merge. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Volumes/Surfaces Selection

Allows you to select volumes or surfaces to use for the new volume.

Operand List

Displays the *Operand List Dialog Box* (on page 89), which displays information about the selected boundaries. You can also use this dialog box to either add or remove volumes or surfaces, or you can rearrange their position in the list.

Finish

Generates the defined volume and exits the command.

Reject Selected Volumes/Surfaces

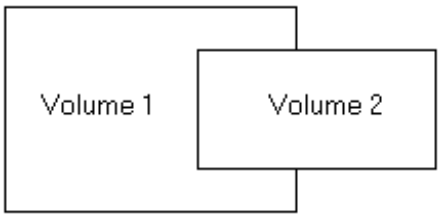
Removes the selected volumes or surfaces from the list and allows you to continue the selection step.

Accept Selected Volumes/Surfaces

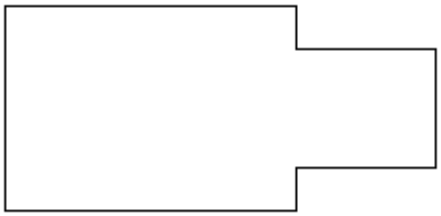
Accepts the selected volumes or surfaces and graphically displays the resulting volume.

Operation

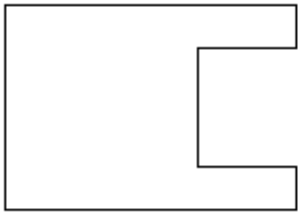
Allows you to specify the operation for selected volume.



- **Add** - Adds the volumes.



- **Subtract** - Subtracts at the intersection of volumes.



- **Intersect** - Creates a volume from the intersection of the volumes.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of volume. Selecting **More** from the dropdown list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

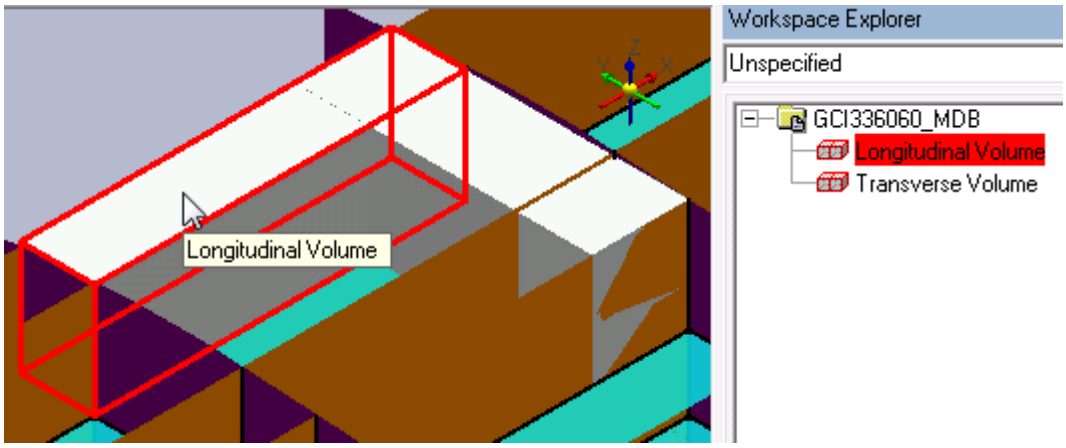
Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Merge multiple volumes

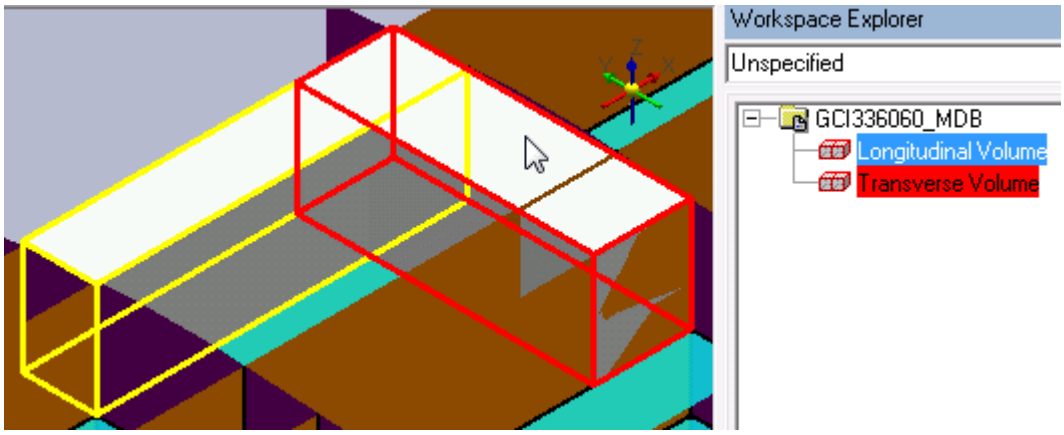
1. On the vertical toolbar, click **Merge Volumes** .
2. Select the first volume to merge.

   TIP You can select the volumes from a graphic view or from the **Workspace Explorer**.



3. Select additional volumes to add.

   TIP You can merge volumes that are adjacent or overlapping.



4. Review the selected volumes, and make any necessary changes using the **Operand List** dialog box.
5. Specify a name for the volume in the **Name** box.
6. Select a volume type from the **Type** list.

TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

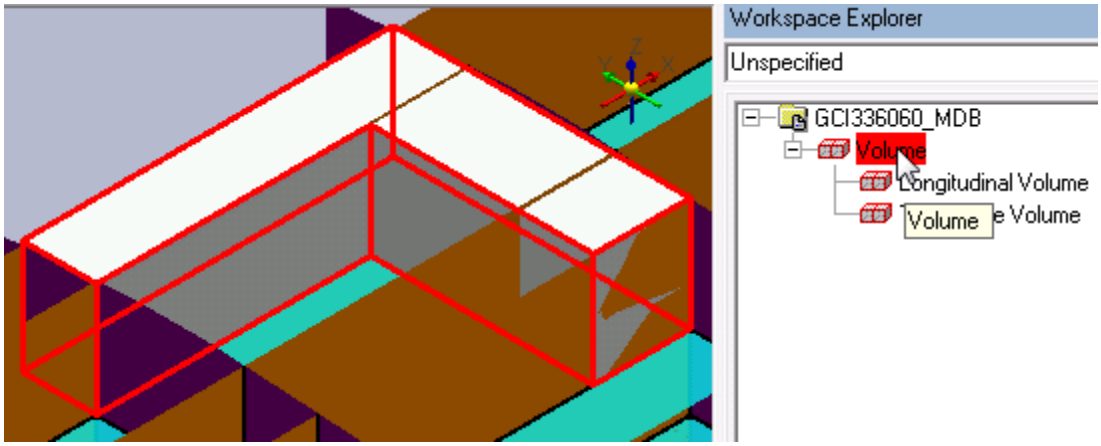
7. Select a folder from the **Space folder** list.

TIPS

- The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
- Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

8. Click **Finish**.

The software creates the volume.



NOTES

- Unlike bound volumes, the act of creating a merged volume is not reversible. If you delete the merged volume, all the volumes that were merged are permanently lost.
- To use a volume as a sub-volume for creation of a merged volume, you must have Full Access permissions on the volume object, and its status must be set to Working.
- You cannot copy or paste a merged volume. You can only delete a merged volume.
- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

Operand List Dialog Box

Sets options for both merging existing volumes using the **Merge Volumes** command and for defining a new volume object using the **Bound by Volumes** command. The **Name** and **Type** properties are read-only.

Name

Displays the name of the volume. You can edit this value on the *Compartment Properties*

Dialog Box (on page 126).

Type

Shows the type of volume. You can edit this value on the **Compartment Volum**e Properties dialog box.

Operation

Specifies whether to add or remove a sub- volume to the current volume definition. You can also specify the intersection between volumes.

Offset

Type or select the offset between the selected plane and a bounding plane. The software creates the bounding plane at an offset from the selected plane when creating the volume.

Add

Allows you to select additional sub-volumes (graphically or from the **Workspace Explorer**) to add to the current bound volume definition. First, select a sub-volume in a graphic view or from the **Workspace Explorer**, and then click **Add** to add it to the list.

Delete

Allows you to select sub-volumes (graphically or the **Workspace Explorer**) and then remove the selected volume from the bound volume definition.

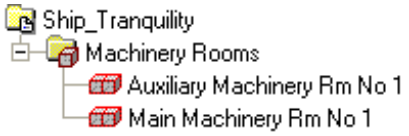
Clear All

Removes all items from the current list. The dialog box remains open, and you can add or remove volumes, as needed.

Bound by Volumes

Defines a volume by selecting existing volumes. This command is particularly useful in placing volumes that cannot overlap. By using existing volumes to define a new volume, you can avoid the problem of creating conflicts among the defined volumes in the model.

The existing volumes remain graphically in the view and in the **Workspace Explorer**.



The sub-volumes that are selected to create the new volume can be selected graphically or from the **Workspace Explorer**. Deleting the new volume deletes the individual integrity of the sub-volumes but does not delete the sub- volumes.

Bound by Volumes Ribbon

Sets options for defining a new volume by specifying sufficient bounding surfaces or volumes.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the compartment, ship zone, or void space that you are placing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Volumes/Surfaces Selection

Allows you to select the volumes and surfaces that serve as boundaries for the new volume.

Operand List

Displays the *Operand List Dialog Box* (on page 89), which displays information about the selected boundaries. You can also use this dialog box to either add or remove volumes, or you can rearrange their position in the list.

Finish

Generates the defined volume and exits the command.

Reject Selected Volumes/Surfaces

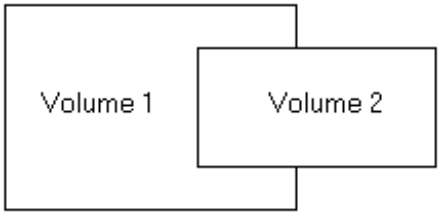
Removes the selected volumes or surfaces from the list and allows you to continue the selection step.

Accept Selected Volumes/Surfaces

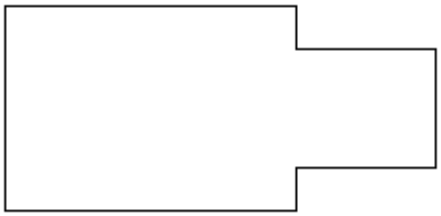
Accepts the selected volumes or surfaces and graphically displays the resulting volume.

Operation

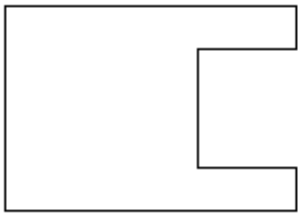
Allows you to specify the operation for selected volume.



- **Add** - Adds the volumes.



- **Subtract** - Subtracts at the intersection of volumes.



- **Intersect** - Creates a volume from the intersection of the volumes.

Name

Displays the default name for the volume (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

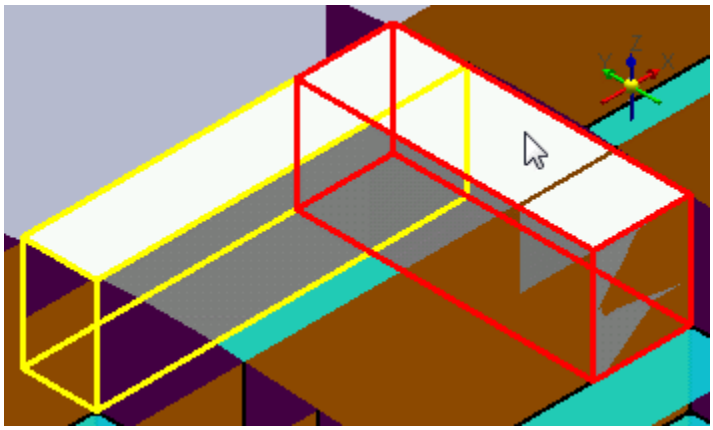
Specifies the type of volume. Selecting **More** from the dropdown list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space folder

Allows you to assign the new volume to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

Place a volume bound by volumes

1. On the vertical toolbar, click **Bound by Volumes** .
2. Select the first volume or surface to use to create the new volume, and then select additional volumes or surfaces.



TIPS

- You can select the volumes or surfaces graphically or from the **Workplace Explorer**.
- The volumes and surfaces that you select can touch or overlap.

3. Review the selected volumes, and make any necessary changes using the **Operand List** dialog box.
4. Specify a name for the volume in the **Name** box.
5. Select a volume type from the **Type** list.

TIP The list in the **Type** box provides only the last few volumes selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of volume found in the catalog database.

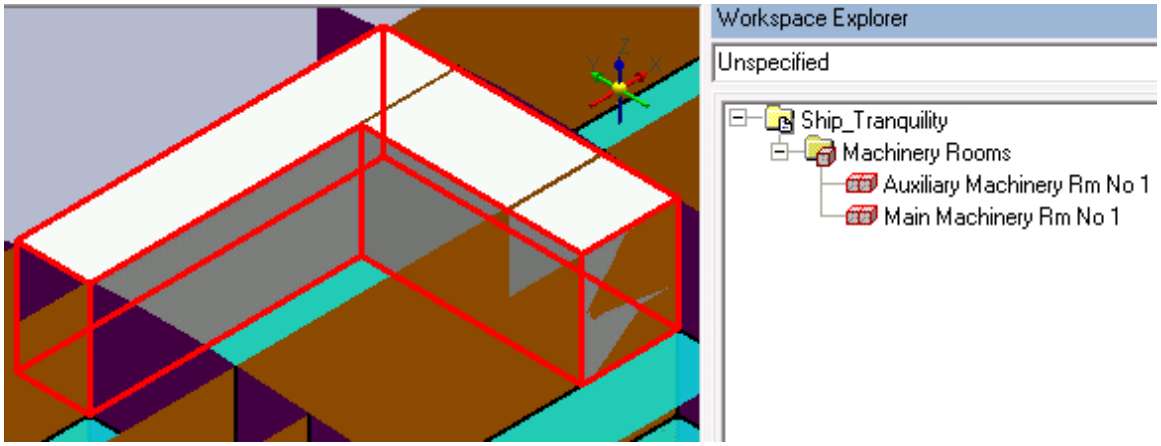
6. Select a folder from the **Space folder** list.

TIPS

- The default folder is the last folder used when creating a volume. In a new session file, the default is the model (the root level of the space hierarchy).
- Assigning volumes to a space folder organizes volumes in the current model. For more information, see *Create Space Folder* (on page 16).

7. Click **Finish**.

The software creates the volume.



NOTES

- The combined volumes retain their individual identities on the **Space** tab of the **Workspace Explorer**.
- You can separate the combined volumes by deleting the bound volume.
- To use a volume as a sub-volume for creation of another volume, you must have full access permissions on the space object, and its status must be set to Working.
- You can click **Volume Properties** on the ribbon to modify the volume properties. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).
- The **Type** list is populated by the reference data and can be customized on a model-by-model basis. For more information on customizing compartmentation reference data, see *Compartmentation Reference Data* in the *Compartmentation Reference Data Guide.*

SECTION 8

Loads

In the Compartmentation task, you can define operating conditions for a hull by assigning loads to volume objects. For example, you can create spatial (volume) loads, unit (point) loads, and combinations of these types of loads.

As with the space objects that you create, you can also create folders to provide a hierarchical storage system for the load objects that you create.

NOTE This functionality is similar to that found in the Structural Analysis task. For more information, see the *Structural Analysis User's Guide*.

Create Load Folder (on page 94)

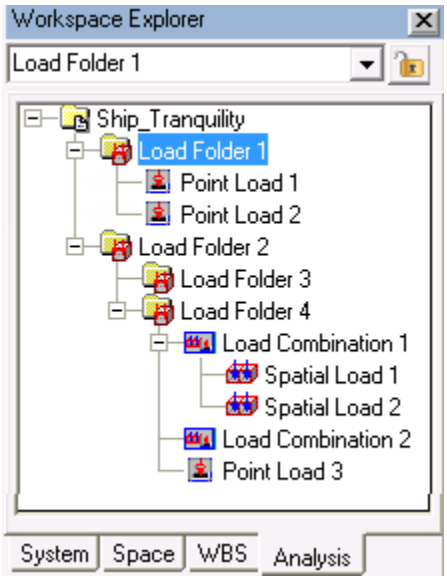
Create a Load Combination (on page 97)

Create a Spatial Load (on page 101)

Create a Unit Load (on page 103)

Create Load Folder

Creates a new hierarchical folder for organizing the load objects that you create in the Compartmentation task. You define a parent and name for the new folder. After the load folder is created, the load folder icon appears on the **Analysis** tab in the **Workspace Explorer**.



The **Analysis** tab of the **Workspace Explorer** displays all load folders, load combinations, and loads in the workspace in a classification hierarchy reflecting the relationships defined for the

load objects. However, an exception is unit loads. A unit load does not appear under a load combination in the **Workspace Explorer** even if it has been added to that load combination. The unit load appears under its load folder.

When you select a load object in the **Workspace Explorer**, the software defaults to edit mode and displays a ribbon to modify the selected load object.

TIPS

- To view the load folders in the **Workspace Explorer**, click **Tools** > **Options**, and select the **Analysis** option on the **General** tab. You also need to make sure the analysis objects are in your workspace definition filter in order to view them in the **Workspace Explorer**.
- You can also right-click a folder on the **Analysis** tab, and then select **Create Load Folder**.

Create Load Folder Ribbon

Sets options for creating a new load folder. After you have completed the required fields, you can click the **Analysis** tab in the **Workspace Explorer** to view the results.

Properties

Displays the **Load Folder Properties** dialog box, which allows you to set properties for the load folder that you are placing. For more information, see *Spatial Load Folder Properties Dialog Box* (on page 154).

Finish

Creates the load folder with the **Name** and **Parent** properties that you specify.

Name

Specifies a name for the new load folder.

Parent

Specifies a location within the load hierarchy. The load folder becomes a child of the selected parent. Selecting **More** in the dropdown list opens the **Select Load Folder** dialog box, which displays the root ship and all load folders. For more information, see *Select Load Folder Dialog Box* (on page 96).

Create a load folder

1. Click **Create Load Folder** on the vertical toolbar.

   TIP You can also right-click a folder on the **Analysis** tab of the **Workspace Explorer**, and then select **Create Load Folder**.

2. On the ribbon, type a name for the new load folder.
3. Specify a parent for the new load folder in the **Parent** list.

   TIP You can select **More** in the drop-down list to display the *Select Load Folder Dialog Box* (on page 96), which displays the full hierarchy.

4. Click **Finish**.

NOTES

- You can click **Properties** on the ribbon to modify the load folder properties. For more information, see *Spatial Load Folder Properties Dialog Box* (on page 154).

Select Load Folder Dialog Box

Specifies the parent load folder needed for determining placement of a new item within the load hierarchy. This dialog box appears when you select **More** in the **Parent** box on the **Create Load Folder** ribbon. By browsing through the load hierarchy, you can select a parent for the new load item. After you select the parent, the software returns you to the model, where you can finalize the creation of the new item.

Look In - Specifies from which location you want the software to pull hierarchical information. You can retrieve hierarchical information from either the current workspace or from the entire Model database.

NOTES

- This dialog box also appears when you select **More** in the **Load folder** box on the **Create Load Combination** and **Create Unit Load** ribbons.
- A similar dialog box, titled **Select Load Combination**, appears when you click **More** in the **Load combination folder** box on the **Create Spatial Load** ribbon. For more information, see *Create Spatial Load* (on page 100).

See Also

Create Load Folder (on page 94)
Create a Load Folder (on page 95)

Create Load Combination

Creates a new load combination, which can consist of spatial loads and unit loads.

A load combination does not appear in a graphic view. In the **Workspace Explorer**, a load combination is represented by an icon on the **Analysis** tab.

NOTES

- To view the load folders in the **Workspace Explorer**, click **Tools** > **Options**, and select the **Analysis** option on the **General** tab. You also need to make sure the analysis objects are in your workspace definition filter in order to view them in the **Workspace Explorer**.
- You can create new loads while running the **Create Load Combination** command. To do this operation, use the **Load List** control.

Create Load Combination Ribbon

Displays options for creating and modifying load combinations.

Properties

Displays the **Spatial Load Combination Properties** dialog box, which allows you to set properties for the load combination that you are placing. For more information, see *Spatial Load Combination Properties Dialog Box* (on page 155).

Spatial Load

Allows you to add or remove a spatial load.

Point Load

Allows you to add or remove a point load.

Load List Control

Displays the **Load List Control** dialog box, which displays information about the selected loads. You can use this dialog box to add or remove loads, as well as create a new load. For more information, see *Load List Control Dialog Box* (on page 99).

Finish

Generates the defined load combination and exits the command.

Reject

Removes the selected load from the **Load List**.

Name

Displays the default name for the load combination (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of load combination. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Load Folder

Allows you to assign the new load combination to a folder in the analysis hierarchy. Selecting **More** from the list opens the *Select Load Folder Dialog Box* (on page 96), which displays the root ship and all load folders. For information about creating load folders, see *Create a load folder* (on page 95).

What do you want to do?

- *Create a load combination* (on page 97)
- *Add a load to a load combination* (on page 98)
- *Add loads using Load List Control* (on page 98)
- *Remove a load from a load combination* (on page 99)
- *Delete a load combination* (on page 99)

Create a load combination

1. Click **Create Load Combination** on the vertical toolbar.
2. Select volumes with applied spatial loads.
3. On the ribbon, click **Point Load**, and click applied point loads.
4. In the **Name** box, type a name for the load combination.
5. Select a type from the **Type** list.

   TIP The list in the **Type** box provides only the last few types selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of load definition found in the catalog database.

6. Select a folder from the **Load folder** list.

   TIPS

   - The default folder is the last folder used when creating a load combination. In a new session file, the default is blank, and you must specify a folder.
   - Assigning load combinations to a load folder organizes loads in the current model. Folders can be created with *Create Load Folder* (on page 94) .

7. Click **Finish**.

NOTES

- If one or more of the loads in the load combination does not have a type assigned, the software notifies you.

Add a load to a load combination

1. Click **Select** on the vertical toolbar.
2. Select **Load Combination** in the **Locate Filter** box.
3. Select the **Analysis** tab in the **Workspace Explorer**.

   TIP To display the **Analysis** tab, click **Tools** > **Options**, and select the **Analysis** check box.

4. Select a load combination on the **Analysis** tab.
5. On the ribbon, click **Spatial Load** to add a spatial load.
6. On the ribbon, click **Point Load** to add a point load.
7. Click **Finish**.

NOTES

- You can also use the **Load List Control** on the ribbon to add and remove loads. For more information, see *Add loads using Load List Control* (on page 98).
- A point load does not appear under a load combination in the **Workspace Explorer** even if it has been added to that load combination. The point load appears under its load folder.

Add loads using Load List Control

The following procedure applies when you are creating load combinations.

1. On the ribbon, click **Load List Control** .
2. Click compartments in a graphic view or in the **Workspace Explorer**. The software adds the compartments and their associated loads to the **Load List Control** dialog box.
3. Select a compartment on the dialog box, and click **Remove** if you change your mind about including that compartment.
4. Click **OK** on the **Load List Control** dialog box to return to the command.

NOTE You must specify the type (description) of each load. If you select a compartment without an associated load, you must specify the description before the software creates the load combination.

Remove a load from a load combination

1. Click **Select** on the vertical toolbar.
2. Select **Load Combination** in the **Locate Filter** box.
3. Select the **Analysis** tab in the **Workspace Explorer**.

   TIP To display the **Analysis** tab, click **Tools** > **Options**, and select the **Analysis** check box.
4. Select a load combination on the **Analysis** tab.
5. On the ribbon, click **Spatial Load** to remove a spatial load.
6. On the ribbon, click **Point Load** to remove a point load.
7. Click **Finish**.

NOTE You can also use the **Load List Control** on the ribbon to remove loads.

Delete a load combination

1. Click **Select** on the vertical toolbar.
2. Select **Load Combination** in the **Locate Filter** box.
3. Select the **Analysis** tab in the **Workspace Explorer**.

   TIP To display the **Analysis** tab, click **Tools** > **Options**, and select the **Analysis** check box.
4. Select a load combination, and then click **Edit > Delete**.

NOTE You can also right-click a load combination, and select **Delete** on the shortcut menu.

Load List Control Dialog Box

Sets options for defining a new load combination using the **Create Load Combination** command.

Compartment Load Tab (Load List Control Dialog Box) (on page 100)
Point Load Tab (Load List Control Dialog Box) (on page 100)

Add Load

Allows you to select additional loads (graphically or from the **Workspace Explorer**) to add to the current load definition. First, select a load in a graphic view or from the **Workspace Explorer**, and then click **Add Load** to add it to the list.

Remove Load

Allows you to select loads (graphically or the **Workspace Explorer**) and then remove the selected load from the load definition.

Clear All

Removes all items from the current list. The dialog box remains open, and you can add or remove loads, as needed.

Attributes

Allows you to specify which attributes appear on the grid. You can choose from **Show All**,

Show Common, or **Show None**.

See Also

Create a Load Combination (on page 97)
Delete a Load Combination (on page 99)
Edit Load Combination Properties (on page 106)
Remove a Load from a Load Combination (on page 99)

Compartment Load Tab (Load List Control Dialog Box)

Sets options for defining compartment loads that make up a load combination.

Compartment

Displays the name of a compartment.

Load Definition

Shows the load definition from the catalog database.

Description

Displays text describing the compartment load.

Load Name

Displays the name of a load on the compartment.

Point Load Tab (Load List Control Dialog Box)

Sets options for defining point loads that make up a load combination.

Load Name

Displays the name of a load on the compartment.

Load Definition

Shows the load definition from the Catalog database.

Description

Displays text describing the point load.

Create Spatial Load

 Places a load distributed through a volume. You can specify the type of material, or cargo, in the volume.

A spatial load appears in a graphic view as a volume. In other words, the graphics of a spatial load are the same as the load's associated volume. In the **Workspace Explorer**, a spatial load is represented by an icon  on the **Analysis** tab.

NOTES

- To display the **Analysis** tab, click **Tools** > **Options**, and select the **Analysis** check box.
- A spatial load can exist only as a child of a load combination. You should create a load combination before creating a spatial load. For more information about creating load combinations, see *Create a load combination* (on page 97).

Create Spatial Load Ribbon

Displays options for creating and modifying a spatial load.

Properties

Displays the **Spatial Load Properties** dialog box, which allows you to set properties for the load that you are placing. For more information, see *Spatial Load Properties Dialog Box* (on page 156).

Spatial Load

Allows you to add or remove compartments for the spatial load.

Finish

Generates the defined spatial load and exits the command.

Reject

Removes the selected load from the **Load List**.

Name

Displays the default name for the load (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of load. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Load Combination Folder

Allows you to assign the load to a load combination folder in the analysis hierarchy. Selecting **More** in the list opens the **Select Load Combination** dialog box, which displays the root ship and all load combination folders. For information about creating load combinations, see *Create a load combination* (on page 97).

Create a spatial load

1. Click **Create Spatial Load** on the vertical toolbar.
2. Click a volume to bear the spatial load.
3. In the **Name** box, type a name for the load.
4. Select a type from the **Type** list.

   TIP The list in the **Type** box provides only the last few types selected from the catalog. Choosing **More** displays the **Select Volume** dialog box, which allows you to select any type of load definition found in the catalog database.
5. Select a load combination from the **Load combination folder** list.

   TIPS

   - The default load combination is the last load combination used when creating a spatial load. In a new session file, the default is blank, and you must specify a load combination.

- Assigning spatial loads to a load combination organizes the loads in the current model. Additional load combinations can be created with the *Create Load Combination* (on page 96) command.

6. Click **Finish**.

NOTES

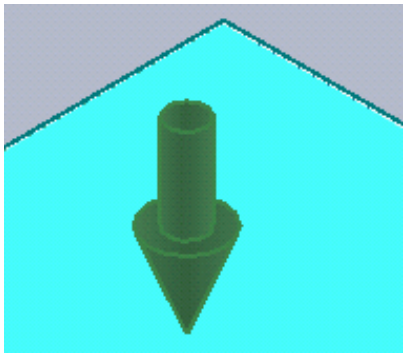
- You can click **Properties** on the ribbon to modify the load properties. For more information, see *Spatial Load Properties Dialog Box* (on page 156).
- To view a spatial load in the **Workspace Explorer**, click **Tools > Options**, and select the **Analysis** option on the **General** tab. You also need to make sure the analysis objects are in your workspace definition filter in order to view them in the **Workspace Explorer** or in a graphic view.

Create Unit Load

Places a load concentrated at a point on a volume.

In the **Workspace Explorer**, a unit load displays as an icon on the **Analysis** tab.

In a graphic view, a unit load appears as an arrow. You might need to zoom in to see the arrow.



In the **Workspace Explorer**, a unit load displays as an icon on the **Analysis** tab.

NOTES

- To display the **Analysis** tab, click **Tools** > **Options**, and select the **Analysis** check box.
- You must create at least one load folder before creating a unit load. For more information, see *Create a load folder* (on page 95).

Create Unit Load Ribbon

Displays options for creating and modifying a unit load.

Properties

Displays the **Unit Load Properties** dialog box, which allows you to set properties for the load that you are placing. For more information, see *Unit Load Properties Dialog Box* (on page 158).

Point Load

Allows you to add or remove points for the unit load.

Finish

Generates the defined unit load and exits the command.

Direction

Select the direction in which the load magnitude is to be applied.

X

Specifies the X-coordinate of the point.

Y

Specifies the Y-coordinate of the point.

Z

Specifies the Z-coordinate of the point.

Value

Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Direction** box.

Name

Displays the default name for the load (as dictated by the active name rule), and allows you to type a different name, if needed.

Type

Specifies the type of load. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Load Folder

Allows you to assign the load to a folder in the analysis hierarchy. Selecting **More** in the list opens the *Select Load Folder Dialog Box* (on page 96), which displays the root ship and all load folders. For information about creating load folders, see *Create a load folder* (on page 95).

Create a unit load

1. Click **Create Unit Load** on the vertical toolbar.
2. Click a point in the model.
3. In the **Direction** box, select a direction for the load applied at the point.
4. In the **Value** box, type the magnitude of the force.
5. In the **Name** box, type a name for the load.
6. Select a type from the **Type** list.

   TIP The list in the **Type** box provides only the last few types selected from the catalog. Choosing **More** displays the *Select Volume Dialog Box* (on page 29), which allows you to select any type of load definition found in the catalog database.

7. Select a folder from the **Load folder** list.

TIPS

- The default folder is the last folder used when creating a unit load. In a new session file, the default is blank, and you must specify a folder.
- Assigning loads to a load folder organizes loads in the current model. Folders can be created with the *Create Load Folder* (on page 94) command.

8. Click **Finish**.

NOTES

- A unit load does not appear under a load combination in the **Workspace Explorer** even if it has been added to that load combination. The unit load appears under its load folder.
- To see the unit load arrow in a graphic view, you may need to fit the view or use other view manipulation commands.
- You can click **Properties**  on the ribbon to modify the load properties. For more information, see *Unit Load Properties Dialog Box* (on page 158).

SECTION 9

Modify Volume Objects

Selecting a placed volume object in your model automatically puts you in editing mode and displays an ribbon that you can use to perform basic editing functions, such as changing the name or volume object type. Using the **Volume Properties** dialog box, you can adjust any of the occurrence properties of the volume.

You can also change the location of a volume within the model using **Move**. For more information, see *Move* in the *Common User's Guide*.

You can change the size of a volume and the location of volume faces using the options used to create the volume. For more information, see each volume command.

The object status and your permissions can affect whether or not you can modify or delete the volume object. A volume that is set to **Approved** cannot be modified or deleted. However, you may be able to modify or delete a volume that is set to **Working**.

Multiple Volumes Edit Ribbon

Sets options for modifying multiple volumes of different construction. This ribbon appears, for example, if you select a volume created by points and a volume created by faces.

Volume Properties

Displays the **Properties** dialog box, which allows you to set properties for the volumes that you are editing. For more information, see *Compartment Properties Dialog Box* (on page 126), *Ship Zone Properties Dialog Box* (on page 138), or *Void Space Properties Dialog Box* (on page 146).

Type

Specifies the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Space Folder

Allows you to assign the volumes to a folder in the space hierarchy. For information about creating space folders, see *Create a space folder* (on page 17).

What do you want to do?

- *Delete a volume object* (on page 106)
- *Edit volume properties* (on page 106)
- *Edit load combination properties* (on page 106)
- *Edit load properties* (on page 107)

Delete a volume object

1. On the vertical toolbar, click **Select** .
2. In the model, select a volume object.

   TIP You can also select a volume object by clicking its name on the **Space** tab in the **Workspace Explorer**.
3. Click **Edit > Delete**.

   TIP You can also click **Delete** on the horizontal toolbar.

NOTES

- The **Delete** command propagates down the hierarchy. For example, if you delete a space folder, its children (if any) and all the volumes contained in the folder are deleted as well.
- Deleting a merged volume deletes all of the sub-volumes associated with it. For more information about merged volumes, see *Merge Volumes* (on page 86).
- Deleting a volume that is bound by other volumes deletes the individual integrity of the sub-volumes but does not delete the sub-volumes themselves. For more information about bound volumes, see *Bound by Volumes* (on page 90).

Edit volume properties

1. On the vertical toolbar, click **Select** .
2. In the model, click the volume object with the properties that you want to edit.

   TIP You can also select a volume object by clicking its name on the **Space** tab in the **Workspace Explorer**.
3. Click **Edit > Properties**.

   TIP You can also click **Volume Properties** on the ribbon to edit properties.
4. On the **Properties** dialog box, click the **General** tab to edit general information about the selected object.

NOTES

- You can click the other tabs on the **Properties** dialog box to edit other properties such as configuration and notes or to traverse relationships among objects.
- You can also modify the properties on the ribbon that appears when you select the object.

Edit load combination properties

1. Click **Select** on the vertical toolbar.
2. Select **Load Combination** in the **Locate Filter** box.
3. In the **Workspace Explorer**, click the **Analysis** tab.

   TIP If the **Analysis** tab is not displayed, click **Tools** > **Options**, and set the tab to display.
4. Select the load combination to edit.

5. Click **Edit > Properties**.

NOTE You can also click **Properties** on the ribbon to edit properties.

Edit load properties

1. Click **Select** on the vertical toolbar.
2. Select **Spatial Load** or **Unit Load** in the **Locate Filter** box.
3. In the **Workspace Explorer**, click the **Analysis** tab.

   TIP If the **Analysis** tab is not displayed, click **Tools** > **Options**, and set the tab to display.
4. Select the load to edit.
5. Click **Edit > Properties**.

NOTE You can also click **Properties** on the ribbon to edit properties.

SECTION 10

Update Attributes

Tools > **Update Attributes** updates custom attributes for compartments and zones. This command calls Visual Basic code to calculate the attribute values.

You can choose to update custom attributes on compartments at any time. This command requires that you have defined custom programming to calculate the attribute values, and that you have bulk loaded a custom attributes workbook in the reference data.

NOTE For more information about the delivered custom attribute data, see *Compartmentation Generic Services* in the *Compartmentation Reference Data Guide*.

Update Attributes Ribbon

Sets options for updating custom attributes on compartments and zones.

Select Compartments or Folders

Allows you to select compartments or zones in a graphic view or in the **Workspace Explorer**. You can also select folders in the **Workspace Explorer** on the **Space** tab.

Finish

Starts the update process.

Computing Attributes

Displays the status of the update. This option only displays after you click **Finish**.

Cancel

Stops the attribute update. This option only displays after you click **Finish**.

Update compartment attributes

1. Click **Tools** > **Update Attributes**.
2. Select compartments and zones in a graphic view or in the **Workspace Explorer** on the **Space** tab.

   TIP You can also select a space folder in the **Workspace Explorer**. The update process will be run on the children volumes contained in that folder.
3. Click **Finish**.

   *The **Computing Attributes** status bar displays. The software updates the attributes on the selected volumes.*

   TIP You can stop the update process by clicking **Cancel**.

SECTION 11

Export Compartment

The **File** > **Export** > **Compartment** command exports a compartment solid body as a .sat file. The .sat file generated by this command can then be imported into a third-party application

NOTE This command is only available in the Compartmentation task, when you have selected a single compartment.

Export a compartment

1. In the Compartmentation task, select a compartment.
2. Click **File** > **Export** > **Compartment**.
3. Browse to the location where you want to save the .sat file.
4. Name the file.
5. Click **Save**.

SECTION 12

Export Compartment as IGES

The **File** > **Export** > **Compartment as IGES** command exports a compartment solid body as an .igs file. The .igs file generated by this command can then be imported into a third-party application.

NOTE This command is only available in the Compartmentation task when you have selected a compartment.

Export Compartment as IGES Ribbon

Select

Allows you to select the compartments.

Finish

Exports the compartments as IGES files in the selected location.

Reject

Rejects the selected compartments.

Accept

Accepts the selected compartments.

Browse

Allows you to browse and save the IGES files.

Export a compartment as IGES

1. Click **File** > **Export** > **Export as IGES**.
2. Click **Select**, and then select a compartment.

   NOTE You can also select the compartments to export before clicking the **Export as IGES** command. The selected compartments are automatically added.
3. Click **Browse**, and browse to the location where you want to save the files.
4. Click **Accept**
5. Click **Finish**.

NOTES

- This command creates an IGES file for each compartment with its compartment name.
- If an IGES file already exists in a location with the same compartment name, then the command adds a suffix to that name. For example, if you have a compartment Tank that already exists in a location, the command creates a Tank_1.igs file.

SECTION 13

Assign Design Parent to Compartments and Volumes

Tools > **Assign Design Parent** allows you to specify the design parent or space parent for compartments and volumes.

Assign Design Parent Ribbon

Select Volume

Allows you to select volumes and compartments, using the **Workspace Explorer** or the graphic view.

Space Parent

Specifies a space folder to use as the space parent of the selected compartments and volumes. Click **More...** to open the **Select Space Folder** dialog box. For more information, see *Select Space Folder Dialog Box (on page 17) in the Smart 3D Compartmentation User's Guide.*

Design Parent

Specifies a design equipment folder to use as the design parent of the selected compartments and volumes. Click **More...** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Smart 3D Common User's Guide*.

Finish

Assigns the selected compartments and volumes to the selected folders.

Assign a space and design parent to compartments and volumes

1. Click **Tools > Add Design Parent**.

   *The **Add Design Parent** ribbon bar displays.*

2. Click **Select Volume**, and then select the compartments and volumes from the **Workspace Explorer** or from the graphic view.
3. Under **Space Parent**, select the space folder to use as the space parent.
4. Under **Design Parent**, select the design equipment folder to use as the design parent.

   NOTE The design parent must be a design equipment folder.

5. Click **Finish**.

SECTION 14

Query Service Command

Provides queries on compartments, space volumes, and blocks. These queries are related to volumes, objects, and geometry. You can export volume and object query results to an .xls or .pdf file. You can save a .sat or .iges file from the geometry queries. Select **Tools > Query Service** to run this command. The queries fall into the following categories:

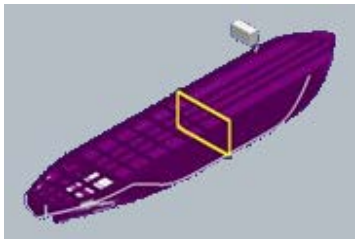
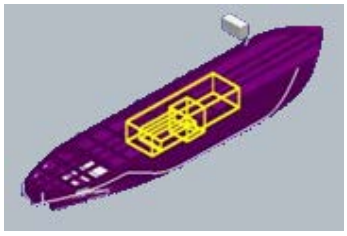
- Volumes
- Objects
- Geometry
- Custom Query

You specify the type of query on the ribbon for this command.

Volume Queries

You can query on compartments, space volumes, and blocks in the model. This type of query is useful, for example, if you want to identify any combination of compartments, space volumes, and blocks that are on or cross a deck or plane.

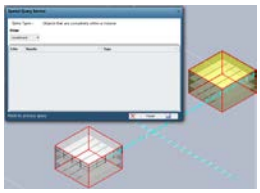
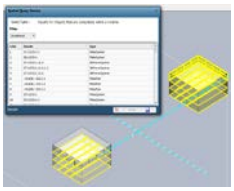
The following table shows a volume query to identify all compartments that cross a plate:

Input	Output
	

Object Queries

These queries allow you to find objects that are fully and partially within a spatial volume.

The following table shows an object query to identify all objects completely within a spatial volume:

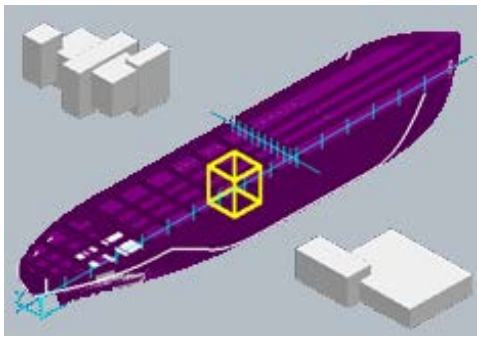
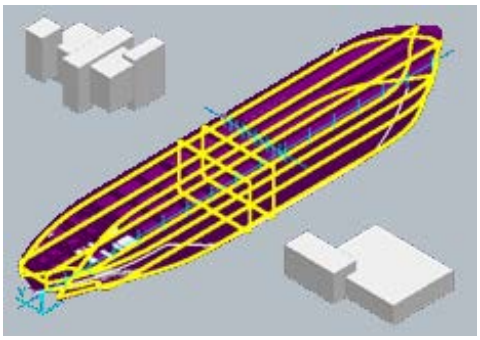
Input	Output
	

Geometry Queries

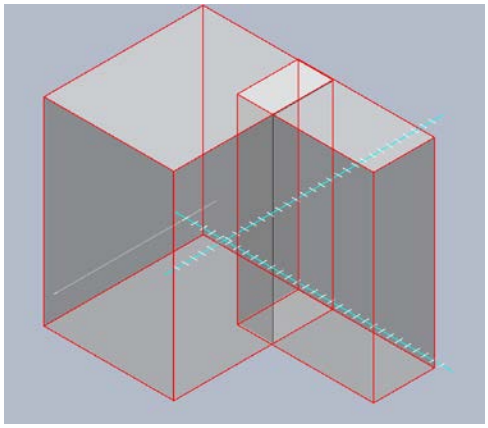
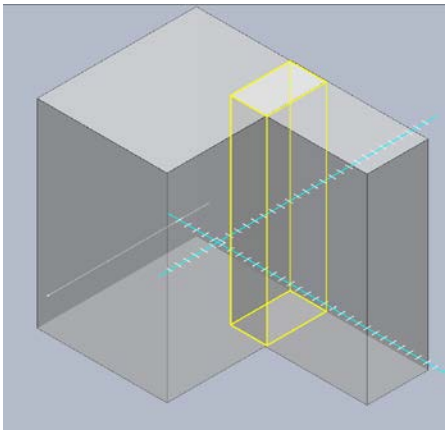
Examples of geometry queries include: finding bounded plates, creating a collection of the faces of a compartment, computing the area and properties of a plane intersecting a compartment, computing the overlap and adjacency between two compartments, and finding the range of a compartment.

An added benefit of the geometry queries is that you can save a .sat or .iges file of the query results.

The following table shows a geometry query to find topological boundaries (bounded plates).

Input	**Output**
	

The following table shows a geometry query to find the overlap geometry between adjacent volumes:

Input	**Output**
	

Custom Query

Runs customized queries on spatial volumes, objects, and geometry. The **Query Service (Custom Query)** ribbon displays options for running customized queries.

Query Service Ribbon

Displays options for running queries on compartments, objects, geometry, and loads.

Query Types

Choose the overall type of query. You can choose from **Volumes**, **Objects**, **Geometry**, or **Custom Query**. Selecting **Custom Query** opens the **Query Service (Custom Query)** ribbon, so that you can specify more options.

Result Type

Specifies the types of objects that you want the query to examine. You can choose from any combination of objects, compartments, or spaces.

 **Reject**

Rejects the selected workspace filter objects.

Finish

Generates and displays the query results.

 **Save**

Saves results of a geometry query to a .pdf, .xls, .sat, or .iges file.

Query Service (Custom Query) Ribbon

Displays options for running customized queries on compartments. This ribbon displays when you select **Custom Query** in the **Query Type** box on the **Query Service** ribbon.

Result Type

Specifies the types of objects that you want the query to examine. You can choose from all objects, compartments, or spaces.

Filter

Specifies a filter to control the results of the query. Click **More** to display the **Select Filter** dialog box.

Direction

Describes the direction relative to the ship.

Query Criteria

Describes the select criteria. For example, the selected criteria can be inside, outside, or overlapping the space volume.

Compart Query Required

Processes the query data with respect to the construction knowledge available to the input object.

 **Reject**

Rejects the selected workspace filter objects.

Finish

Generates the query results.

 **Save**

Saves results of a geometry query to a .pdf, .xls, .sat, or .iges file.

Add To Select Set

Adds the query results to the select set.

What do you want to do?

- *Run a spatial query* (on page 115)
- *Run a custom query* (on page 115)

Run a spatial query

1. Click **Tools** > **Query Service**.

   *The **Query Service** ribbon and dialog box display.*
2. On the **Query Service** ribbon, select a type of query in the **Query Types** box.
3. Set **Result Type** to the types of objects to query.
4. Select objects in a graphic view or in the **Workspace Explorer**.

   TIP The message in the status bar indicates the type of object to select.
5. Click **Finish**.

   *The query results display in the **Query Service** dialog box.*

NOTES

- You can select the results in the **Query Service** dialog box to highlight the corresponding object in red in the graphic view.
- For a geometry query, click **Save** to save the results to a .sat or .iges file.

Run a custom query

1. Click **Tools** > **Query Service**.

   *The **Query Service** ribbon and dialog box display.*
2. On the **Query Service** ribbon, set **Query Type** to **Custom Query**.

   *The **Query Service (Custom Query)** ribbon displays.*
3. Under **Result Type**, select the type of objects to query.
4. Select objects in a graphic view or in the **Workspace Explorer**.

   TIP The message in the status bar indicates the type of object to select.
5. Specify the other ribbon properties as necessary to define the query.
6. Click **Finish**.

   *The query results display in the **Query Service** dialog box.*

NOTES

- You can select the results in the **Query Service** dialog box to highlight the corresponding object in red in the graphic view.

- For a geometry query, click **Save**  to save the results to a .sat file.

Query Service Dialog Box

Displays the results of the query you ran. This dialog box appears when you click **Tools > Query Service**.

Query Types

Specifies what is being queried. The following types of queries are available:

Volumes

Volumes on a deck - Reports volumes that are on the input deck plate.

Volumes cross a deck - Reports volumes that cross the input deck plate.

Volumes on or cross a deck - Reports volumes that either cross or are on the input deck plate.

Volumes on a plane - Reports volumes that are on the input plane.

Volumes crossing a plane - Reports volumes that cross the input plane.

Volumes on or crossing a plane - Reports volumes that either cross or are on the input plane.

Volumes by Part - Reports volumes by the input volume part name.

Volumes by PartClass - Reports volumes by the input volume part class.

Volume based on point in model - Reports volumes on the given input point.

Volumes connected (get adjacent Volumes) - Reports volumes that are adjacent to the input volume.

Volumes connected on the same deck or plane - Reports volumes that are connected on the input deck plate or plane.

Objects

Objects that are completely within a Volume - Reports objects that are completely inside the input volume.

Objects that cross a Volume - Reports objects that cross the input volume.

Objects that are Inside and Touching a Volume - Reports objects that are completely inside and touching the input volume.

Objects that are Outside and Touching a Volume - Reports objects that are completely outside and touching the input volume.

Objects that are Outside a Volume - Reports objects that are outside the input volume.

Objects Encompass and Touching a Volume - Reports objects that are encompassed within and touching the input volume.

Objects Encompass and within a Volume - Reports objects that are encompassed within the input volume.

Objects whose center point is within a Volume - Reports objects whose center point is within the input volume.

Geometry

Specify Volume topological boundaries (get bounded plates) - Reports topological boundaries (bounded plates) for the input volume.

Get the Volume Geometry - Retrieves the geometry of the input volume, as specified in a .sat file or an .iges file.

Compute the enclosed volume - Computes and displays the enclosed volume of the input volume.

Compute the enclosed volume's centroid below a given height (fill-height is an example) - Computes and displays the centroid of the input volume.

Compute the overlap geometry between two adjacent Volumes - Retrieves the overlap geometry of two adjacent input volumes, as specified in a .sat file or an .iges file.

Compute the overlap geometry Area between two adjacent Volumes - Computes and displays the intersection type and intersection geometry of two adjacent input volumes.

Get the range of a Volume (along global CS) - Computes and displays range points of the input volume along the global coordinate system.

Get the minimum range of a Volume (optimal calculation) - Computes and displays the minimum range points of the input volume.

Get the minimum range of a Volume along Active CS - Computes and displays the minimum range points of the input volume along the global coordinate system.

Get the surfaces of the Volume geometry - Retrieves the geometry surfaces of the input volume, as specified in a .sat file or an .iges file.

Get Overlapping geometry for Volumes on a deck or plane - Retrieves the overlapping geometry between the input deck plate and the volumes on that plate, as specified in a .sat file or an .iges file.

Get Subtracted geometry for two objects - Retrieves the subtracted geometry of the two input volumes, as specified in a .sat file or an .iges file.

Retrieve an attribute on the selected Volume - Reports the attribute of the input volume.

Get Intersection Geometry of Volume and specific type Objects Overlapping Volume - Retrieves the intersection geometry for the input volume and the filter, as specified in a .sat file or an .iges file.

Custom Query

Computes and reports the objects, such as volumes or parts, according to the input.

Filter

Specifies a filter to control the results of the query. Click **More** to display the **Select Filter** dialog box.

Sl No.

Displays sequential numbers corresponding to the returned objects.

Results

Displays the object names.

Type

Displays the object type.

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style Rules**, and other **Select by Filter** commands, including Project Management's **Model Data Reuse** (MDR), Drawings View Styles, and Reports commands that require runtime filter selection. You can access this dialog box in several ways.

- Select **File** > **Define Workspace**, and select the **More** option in the **Filter** box.
- Select **Format** > **Surface Style Rules**, click **New** or **Modify**, and then select the **More** option in the **Filter** box.
- Select **Tools** > **Select by Filter.**

The tree view displays the following types of filters:

- **Catalog Filters** - These filters are used to reference data in the Catalog. For example, a catalog filter could apply to company-wide operations. Your administrator can define Company_Filter_1, Company_Filter_2, and so forth.
- **Model Filters** - These filters are available to everyone assigned to a specific model database. There are delivered catalog filters to query on the different types of model objects. You must have the appropriate privileges to create, edit, or delete these filters.
- **My Filters** - These are personal filters that you create and place in the **My Filters** folder. They are visible only to you, the owner. You cannot see the personal filters of others, and they cannot see your personal filters. Select a filter from one of the listed filters, or create a new filter to meet your specific requirements.

 **New Folder**

Creates a new folder.

 **New Filter (Simple or Asking)**

Displays the **New Filter Properties** dialog box so that you can create a new filter. Asking filters allow you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to supply). The values apply to properties that you have already designated you will supply when the filter runs. Asking filters are portable between models.

NOTE Model Data Reuse (MDR) does not support asking filters. The only valid filter types for an MDR transaction are System, Permission Group, Object Type, Volume and Properties. You can define the filter on any one of these tabs or in a combination using multiple tabs.

 **New Compound Filter**

Displays the **New Compound Filter Properties** dialog box, which you use to create a new compound filter containing the Or, And, or Not operators. Compound filters are not supported for MDR.

 **New SQL Filter**

Displays the **New SQL Filter Properties** dialog box, in which you can type the text of an SQL query. SQL filters are not supported for MDR

Delete

Removes a filter or folder from the **Select Filter** list. If you delete a folder, the software also deletes its contents.

Rename

Changes the name of an existing filter or folder from the **Select Filter** list.

Properties

Displays the **Filter Properties** dialog box so that you can select the properties that determine your filter search criteria.

NOTES

- If this dialog box is activated from the **Select by Filter** command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT, and click each filter. When you click **OK**, all objects that fit the selected filters are selected.
- If this dialog box is activated from the **Select by Filter** command, it clears the select set before adding objects to the select set.

Select by Filter
Select Objects by Filter

SECTION 15

Submit Batch Job

Tools > **Submit Batch Job** runs batch processes using the Intergraph Batch Services framework.

This command runs the following types of batch jobs:

Detailing:

- Detail parts
- Undetail parts
- Update reports

Production:

- Generate manufacturing parts
- Generate XML output files
- Manufacturing Service Manager update

Custom batch processes:

- BO recompute
- Other custom batch processes that you define.

What do you want to do?

Run a detailing batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Detailing** from the **Batch Process Type** list.

   *Smart 3D displays the available detailing batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*
7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a production batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*
2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*
3. Select **Production** from the **Batch Process Type** list.

   *Smart 3D displays the available production batch processes in the **Select Batch Process** list.*
4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*
5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*
7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a custom batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*
2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*
3. Select **Custom Batch PRocesses** from the **Batch Process Type** list.

   *Smart 3D displays the available custom batch processes in the **Select Batch Process** list.*
4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*
5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Add a custom batch process

1. Add the ProgID and job description of the custom batch process to the <Product Folder>\CommonShip\SOM\Client\Xml\CustomBatchJobDetailsConfiguration.xml file.

   For example:

   ```
   <JOB>
      <JobType>CustomBatchProcessD</JobType>
      <JobType_LocaleID>103</JobType_LocaleID>
      <JobDescription>Description of
   CustomBatchProcessD</JobDescription>
      <JobDescription_LocaleID>104</JobDescription_LocaleID>
      <JobProgId> ProgID of CustomBatchProcess </JobProgId>
      <IsModelSpecific>False</IsModelSpecific>
   </JOB>
   ```

2. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

3. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

4. Select **Custom Batch Process** from the **Batch Process Type** list.

   *The available custom batch processes display in the **Select Batch Process** list.*

5. Select the custom process to run, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

6. Specify the **Batch Input Type**, and add inputs as necessary.
7. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

8. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Generic Batch Command Dialog Box

Displays the controls used to submit a generic batch command.

Batch Process

Specifies the batch process to run. The list contains recently-selected job types. Click **More** to display the **Batch Process Selection** dialog box, and select a process that is not on the list. For more information, see *Batch Process Selection Dialog Box* (on page 123).

Batch Input Type

Indicates the input type for the batch process.

- **Filter** indicates that you are providing a filter as an input to the batch job.
- **Assembly** indicates that you are providing an assembly as input to the batch job.
- **Object** indicates that you are providing a set of objects as input to the batch job.

 **Add**

Adds inputs to your batch job.

- If **Filter** is selected, this button displays the **Select Filter** dialog box so that you can add filters as inputs to your batch job.
- If **Assembly** is selected, this button displays the **Select Assembly** dialog box so that you can add assemblies as inputs to your batch job.
- If **Object** is selected, this button is not available. To add objects as inputs, select them from the model or the **Workspace Explorer**.

 **Remove**

Removes the selected inputs from the list. The software only removes the inputs that have the associated check box selected.

Share

Saves the inputs for the next batch process. If this check box is cleared, Smart 3D clears the inputs when you select a different batch process.

Schedule as Multiple Jobs

Processes each item in the inputs list as a separate batch process. If this check box is cleared, Smart 3D processes all of the inputs as a single batch process.

Schedule

Displays the **Schedule [Task]** dialog box so that you can schedule the batch process. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Cancel

Exits the command.

Batch Process Selection Dialog Box

Displays the controls used to specify the type of batch process to run.

Batch Process Type

Displays the general batch process types. Select a type to display the available batch process of that type in the **Select Batch Process** list.

Select Batch Process

Displays the available batch process associated with the selected type.

APPENDIX A

Appendix: Property Dialog Boxes

This appendix contains reference information for all the property dialog boxes in the Compartmentation task.

In This Appendix

Common Property Tabs

The software displays some common property tabs on the **Properties** dialog boxes for all volume and load objects. Instead of repeatedly listing the common tabs for each **Properties** dialog box, they are documented here for easy reference.

Configuration Tab (on page 124)
Relationship Tab (on page 125)
Notes Tab (on page 125)

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Relationship Tab

Displays all objects related to the object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Name

Displays the name of the related object.

Type

Displays the type of related object.

Go To

Displays the properties of the selected object.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, only one of the notes will show on the drawing. It is important to know about and consider this situation when defining notes on an

object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date the note was created. The system automatically supplies the date.

Time

Displays the time the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the logon name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, a dimension can be included for the component in Support drawings if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Compartment Properties Dialog Box

Displays compartment properties for review and editing.

NOTES

- The **Cross-Section** tab only appears for compartments created with the **Place Volume Along Path** command.
- The dialog box name varies based on the **Type** you select while creating the compartment class.

General Tab (Compartment Properties Dialog Box) (on page 127)
Cross-Section Tab (on page 137)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Compartment Properties Dialog Box)

Displays and defines the general properties of the selected compartment.

NOTE Because compartment properties are customizable in the reference data, only the properties that are required by the software are documented.

Compartment properties are divided into several different categories: **Standard**, **Surface Treatment and Coating**, **Spatial Standard**, **Spatial Frames**, **Spatial Tightness**, and **Spatial Structural**. You select which category that you want to define values for by using the **Category** option.

Type

Displays the type of volume. Select **More** from the list to open the *Select Volume Dialog Box* (on page 29), and select a type.

Standard

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Space Folder**

Selects the space folder in which the selected compartment resides. Space folders are used to organize volume objects within the model.

Class

Displays the classification of the volume as a compartment or zone. This field is read-only and updates automatically when you select a type of volume in the **Type** box.

Structure Type

Displays or defines the type of structure for the compartment. If you want to add, edit, or remove values that are available for selection, edit the **Compartment Structure Type** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Function Type

Displays or defines the type of function for the compartment. If you want to add, edit, or remove values that are available for selection, edit the **Compartment Function Type** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Compartment Number

Displays or defines the number for the compartment.

Compartment Group

Displays or defines the grouping for the compartment. If you want to add, edit, or remove values that are available for selection, edit the **Compartment Compartment Group** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Compartment Type

Displays or defines the type of compartment. If you want to add, edit, or remove values that are available for selection, edit the **Compartment Compartment Type** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Relevant Cargos

Displays or defines the cargo that the compartment can carry. If you want to add, edit, or remove values that are available for selection, edit the **Compartment Relevant Cargos** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Filling Height

Specifies the height to which the compartment can be filled.

Water Height in Damaged Condition

Specifies the height to which water can rise in a damaged compartment.

Surface Treatment and Coating

Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog. If you want to add, edit, or remove values that are available for selection, edit the **Compart Coating Type** sheet in the CompartmentCodeLists.xls workbook in the reference data. Smart 3D includes this property in the painting area report.

Coating Level

Specifies a value for the coating level. Smart 3D includes this property in the painting area report.

Coating Status

Specifies the status of the coating for the compartment. This list is defined by the CompartmentCoatingStatus codelist in the CompartmentCodeLists.xls workbook in the reference data.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Coating Recoating

Specifies the recoating condition for the object.

True indicates that the object is recoated.

False indicates that the object is not recoated.

Reference Position

Specifies the position of the compartment with reference to the ship. The list is defined by the ReferencePosition codelist.

Spatial Standard

User Description

Type text that describes the user of the compartment.

User Purpose

Type text that describes the purpose of the compartment.

User Purpose Description

Type text for the user and purpose of the compartment.

Steel Reduction

Type a value for steel reduction.

Permeability

Type a value for permeability.

Tmy

Type the Tmy value, which is the moment of inertia with the waterline at 50 percent filling. The unit for this value is that for the second moment of area, for example, m4.

Capacity

Type a value for capacity.

Spatial Frames

Longitudinal Minimum

Displays the minimum longitudinal value for the compartment.

Longitudinal Maximum

Displays the maximum longitudinal value for the compartment.

Deck Minimum

Displays the minimum deck value for the compartment.

Deck Maximum

Displays the maximum deck value for the compartment.

Transversal Minimum

Displays the minimum transversal value for the compartment.

Transversal Maximum

Displays the minimum transversal value for the compartment.

Spatial Tightness

Compart Tightness

Specifies the type of tightness for the compartment. Examples include air-tight, water-tight, and so on. The list is defined by the CompartTightness codelist. Smart 3D includes this property in the tightness report.

Struct Tightness

Specifies the type of tightness for the surrounding structure. This type is evaluated from the bounded structures of the compartment. The list is defined by the StructPlateTightness codelist. Smart 3D includes this property in the tightness report.

Spatial Structural

Volume Molded

Displays the molded volume of the compartment.

Wall Area

Displays the area of a wall.

Wall Length

Displays the length of a wall.

Deck Height

Displays the height of the deck.

Side Wall Area

Displays the area of the side wall.

Bottom Area

Displays the area of the bottom.

Center of Gravity Bottom X

Displays the X-coordinate for the center of gravity of the bottom.

Center of Gravity Bottom Y

Displays the Y-coordinate for the center of gravity of the bottom.

Center of Gravity Bottom Z

Displays the Z-coordinate for the center of gravity of the bottom.

Dimensions

Center of Gravity X

Displays the X-coordinate for the center of gravity of the volume.

Center of Gravity Y

Displays the Y-coordinate for the center of gravity of the volume.

Center of Gravity Z

Displays the Z-coordinate for the center of gravity of the volume.

Volume

Specifies the volume of the compartment or zone.

Moment of Inertia X

Displays the X-coordinate for the moment of inertia of the volume.

Moment of Inertia Y

Displays the Y-coordinate for the moment of inertia of the volume.

Moment of Inertia Z

Specifies the Z-coordinate for the moment of inertia of the volume.

Surface Area

Specifies the surface area value of the volume.

Range High X

Displays the X-coordinate for the upper end of the volume.

Range High Y

Displays the Y-coordinate for the upper end of the volume.

Range High Z

Displays the Z-coordinate for the upper end of the volume.

Range Low X

Displays the X-coordinate for the lower end of the volume.

Range Low Y

Displays the Y-coordinate for the lower end of the volume.

Range Low Z

Displays the Z-coordinate for the lower end of the volume.

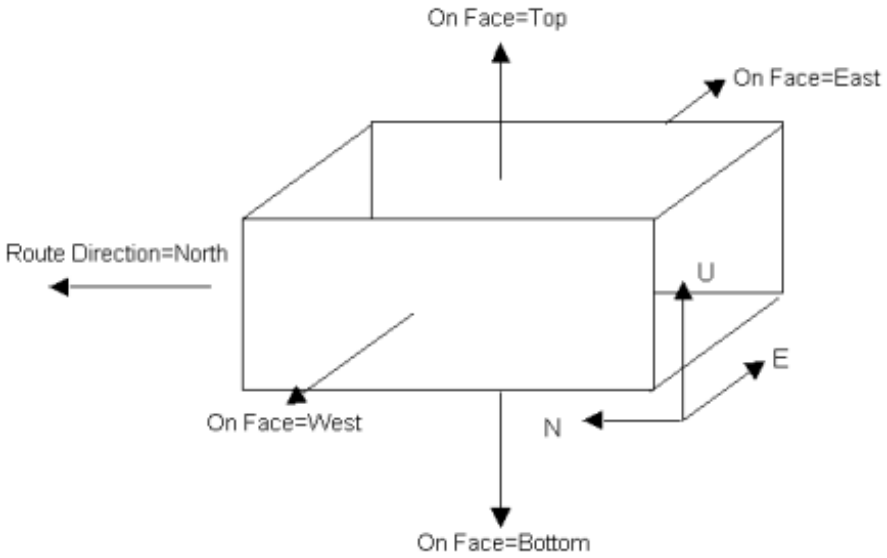
★ IMPORTANT The following route zones will display additional **Standard** category properties specific to their zone type.

Pipe Attraction Zone

The Pipe Attraction Zone models a volume that attracts pipes. It has a cost factor, which means that routes that lie within the Pipe Attraction Zone are less expensive than equivalent routes outside the zone. The Pipe Attraction Zone is similar to the Pipe Rack Zone in that it modifies the result of the routing calculation by attracting pipes but has fewer rules for layout and for joining and leaving the zone.

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Cost Factor

Specifies the cost factor applied to pipe that runs through the Pipe Attraction Zone. Typically, the cost factor value is a number less than one. The lower the number, the more

the zone attracts pipe.

Pipe Avoidance Zone

The Pipe Avoidance Zone models a volume that the pipe can enter if it is essential, but will avoid if possible. For example, it is not good practice for pipe to be placed within the immediate vicinity of a tank, but some of the pipes must connect to nozzles on the tank.

Cost Factor

Specifies the cost penalty applied to pipe that runs in the Pipe Avoidance Zone. Typically, the cost factor value is a number greater than one. The higher the number, the more the zone repels pipe.

Pipe Connection Zone

Pipe Connection Zones define a volume at which a route can terminate. They may act as obstruction zones (as defined by the **Obstruction** property) in addition to their special connection properties.

Connection Face

Specifies on which face the connection is made.

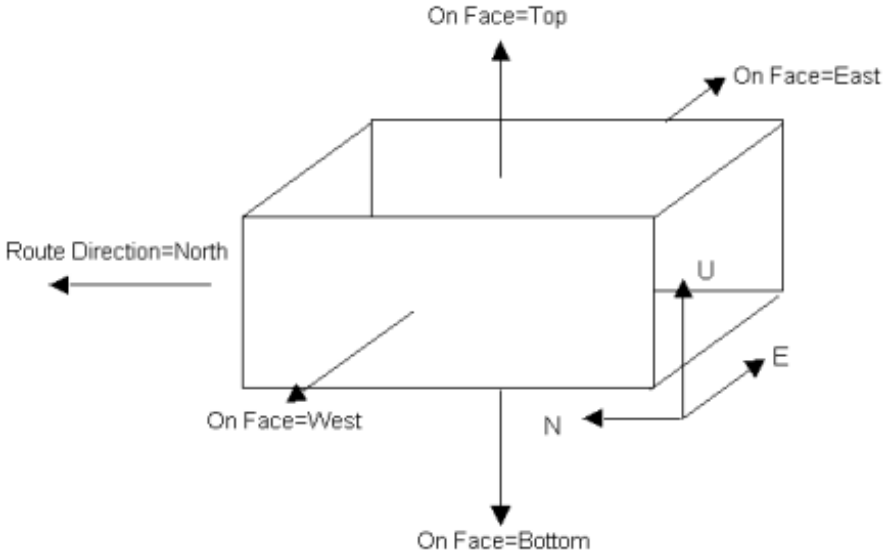
Obstruction

Specifies whether the zone allows pipes to be routed through it (value = **False**) or whether the zone allows pipes to be routed up to it, but not through it (value = **True**).

Pipe Rack Zone

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior

of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Layout

Specifies how pipes are laid out. The software supports five layout types:

At Edge - Pipes are laid out in joining order starting at each edge and working inwards.

Large Bore At Edge - Pipes are laid out in bore order starting at each edge and working inwards.

Spreadout - Pipes are laid out evenly across the rack.

At Center - Pipes are laid out in joining order starting at the center of the rack and working outwards.

Large Bore At Center - Pipes are laid out with the largest bore at the center and then working outwards.

Allow

Filters the pipes that can use the zone based on fluid type. Only the fluid types you specify are permitted on the rack. If more than one fluid type is to be allowed, use a comma or a space to separate them. For example, type **Process,Steam**.

Cost Factor

Displays the cost factor applied to pipe that runs on the rack. The software calculates this value based on data you specify for properties in the **Cost Estimation** category. Typically, the cost factor value is a number less than one. The lower the number, the more the zone attracts pipe.

Custom

Displays all custom properties defined for the volume in the reference data. You can change any of the values on this dialog box both before and after placement.

User Attribute

Displays the name of the custom property defined in the reference data. All custom attributes for the selected volume object appear.

Value

Displays the values for the custom properties of the selected volume object. You can change these values as needed.

IMPORTANT The following category is only displayed when you select either **Pipe Rack Zone** or **Icarus Open Steel Zone** in the **Type** list.

Cost Estimation

Displays properties required by Aspen Icarus Process Evaluator™ to compute a cost that is based on Alias auto- routing.

NOTE Aspen Icarus Process Evaluator is a third-party software product that allows Process Engineers to accurately evaluate the economic impact of their process designs.

Length

Specify the cost estimation length.

Width

Specify the cost estimation width.

Height

Specify the cost estimation height.

Number of Stairways

Specify the number of stairways for cost estimation.

Grating Type

Specify the type of grating used in the zone for cost estimation.

Structural Steel Analysis

Specify the type of structural steel analysis for cost estimation.

Column Base End Condition

Specify the type of connection at the column base for cost estimation.

Distributed Load Per Level

Specify the distributed load per level for cost estimation.

Wind Force Adjustment

Specify the adjustment for wind force for cost estimation.

Seismic Force Adjustment

Specify the adjustment for seismic force for cost estimation.

IMPORTANT The following **Cost Estimation** properties are only available when you select **Pipe Rack Zone** in the **Type** list.

Pipe Rack Type

Specify the type of pipe rack for **Pipe Rack Zone** cost estimation.

Concrete Type

Specify the type of concrete for **Pipe Rack Zone** cost estimation.

Number of Levels

Specify the number of levels for **Pipe Rack Zone** cost estimation.

Height First Level

Specify the height of the first level for **Pipe Rack Zone** cost estimation. The default measurement is in feet and inches.

Number of Ladders

Specify the number of ladders for **Pipe Rack Zone** cost estimation.

Number of Braced Bays

Specify the number of braced bays for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Pipe Level

Specify the number of beam struts on each pipe level for **Pipe Rack Zone** cost estimation.

Number Catwalks

Specify the number of catwalks for **Pipe Rack Zone** cost estimation.

Catwalk Width

Specify the required width of the catwalk for **Pipe Rack Zone** cost estimation.

Main Bent Spacing

Specify the main bent spacing for **Pipe Rack Zone** cost estimation.

Third Column Option

Specify whether or not a third column is necessary for **Pipe Rack Zone** cost estimation. For a two-bay rack, a third column is required.

Air Cooler Loading

Specify the maximum load for the air cooler for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Column Line

Specify the number of beam struts for each column line for **Pipe Rack Zone** cost estimation.

End Bent Exclusion

Specify whether or not to excluded end bents in the zone for **Pipe Rack Zone** cost estimation.

Minimum Beam Or Column Width

Specify the minimum width of the beam or column for **Pipe Rack Zone** cost estimation.

★ IMPORTANT The following **Cost Estimation** properties are only available when you select **Icarus Open Steel Zone** in the **Type** list.

Number of Floors

Specify the number of floors for **Icarus Open Steel Zone** cost estimation.

Bay Span

Specify the span of the bay for **Icarus Open Steel Zone** cost estimation.

Bay Width

Specify the width of the bay for **Icarus Open Steel Zone** cost estimation.

Floor Grate Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Siding Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Floor Slab Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Slab Thickness

Specify the thickness of the slab for **Icarus Open Steel Zone** cost estimation.

See Also

Compartment Properties Dialog Box (on page 126)

Cross-Section Tab

Displays and defines information about the cross-section of the selected compartment or zone.

NOTE This tab is only available for volume objects created with the **Place Volume Along Path** command.

Standard

Cross-Section

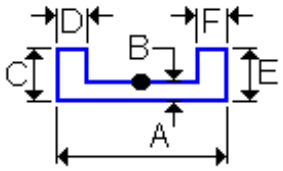
Specifies the type of cross-section for the volume. You can either select a standard cross-section defined in the reference data or select **Sketch** to draw your own cross-section.

When a standard cross-sectional type is selected in the **Cross-Section** box, you can modify the properties that are described later in this topic. When **Sketch** is selected in the **Cross-Section** box, you can view each point, its coordinates, and its turn type in the table, but you cannot edit the data. For more information, see *General Tab (Sketch Properties Dialog Box)* (on page 153).

Display Cross-Section Image

Displays the image associated with the standard cross-section type in the reference data.

Cross-section images show the dimensions that you can define for the cross-section type, the default cardinal points for the cross-section type, and the angle for the cross-section. For example, the following graphic contains the dimensions and cardinal point for a standard road cross-section:



If no image is associated with the cross-section and the selected cardinal point, **Image Not Available** displays at the bottom of the tab.

Letters A, **B**, **C**, and so forth

Defines the dimensions for standard cross-sections. If an image is defined for the cross-section in the reference data, you can see what each letter represents by clicking **Display Cross Section Image**.

Cardinality

Defines the point where the software attaches the cross-section to the path. If you click **Display Cross-Section Image** to see a graphic of the cross-section, you can view where each cardinal point is located by selecting each cardinal point in the list. The software automatically updates the display, if the appropriate graphic is available, with a graphic containing the selected cardinal point.

You can select **User Defined** cardinal points to specify a reference point on selected cross section for precise placement.

Rotation Angle

Specifies the rotation of the cross-section about its local axis.

See Also

Compartment Properties Dialog Box (on page 126)

Ship Zone Properties Dialog Box

Displays ship zone properties for review and editing.

NOTE The **Cross-Section** tab only appears for ship zones created with the **Place Volume Along Path** command.

General Tab (Ship Zone Properties Dialog Box) (on page 138)
Cross-Section Tab (on page 137)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Ship Zone Properties Dialog Box)

Displays and defines the general properties of the selected ship zone.

NOTE Because ship zone properties are customizable in the reference data, only the properties that are required by the software are documented.

Ship zone properties are divided into several different categories: **Standard**, **Surface Treatment and Coating**, and **Dimensions**. You select which category that you want to define values for by using the **Category** option.

Type

Displays the type of volume. Selecting **More** from the list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Standard

Name

Displays or defines the name of the ship zone. The name is based on the **Name Rule** selection. If you want to type a new name, in the **Name Rule** box, select **User Defined**, and then type a name in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this ship zone. You can select one of the listed rules or select **User Defined** to specify the name yourself in the **Name** box.

Space Folder

Selects the space folder in which the selected ship zone resides. Space folders are used to organize volume objects within the model.

Class

Displays the classification of the volume as a compartment or zone. This field is read-only and updates automatically when you select a type of volume in the **Type** box.

Surface Treatment and Coating

Reference Position

Displays the position of the ship zone with reference to the ship.

Dimensions

Center of Gravity X

Displays the X-coordinate for the center of gravity of the volume.

Center of Gravity Y

Displays the Y-coordinate for the center of gravity of the volume.

Center of Gravity Z

Displays the Z-coordinate for the center of gravity of the volume.

Volume

Specifies the volume of the compartment or zone.

Moment of Inertia X

Displays the X-coordinate for the moment of inertia of the volume.

Moment of Inertia Y

Displays the Y-coordinate for the moment of inertia of the volume.

Moment of Inertia Z

Specifies the Z-coordinate for the moment of inertia of the volume.

Surface Area

Specifies the surface area value of the volume.

Range High X

Displays the X-coordinate for the upper end of the volume.

Range High Y

Displays the Y-coordinate for the upper end of the volume.

Range High Z

Displays the Z-coordinate for the upper end of the volume.

Range Low X

Displays the X-coordinate for the lower end of the volume.

Range Low Y

Displays the Y-coordinate for the lower end of the volume.

Range Low Z

Displays the Z-coordinate for the lower end of the volume.

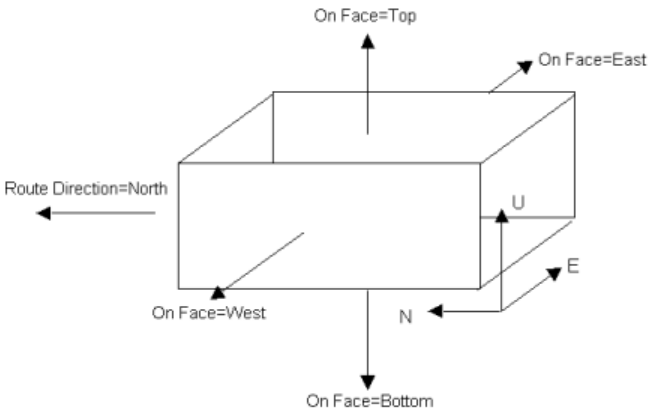
★ IMPORTANT The following route zones will display additional **Standard** category properties specific to their zone type.

Pipe Attraction Zone

The Pipe Attraction Zone models a volume that attracts pipes. It has a cost factor, which means that routes that lie within the Pipe Attraction Zone are less expensive than equivalent routes outside the zone. The Pipe Attraction Zone is similar to the Pipe Rack Zone in that it modifies the result of the routing calculation by attracting pipes but has fewer rules for layout and for joining and leaving the zone.

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Cost Factor

Specifies the cost factor applied to pipe that runs through the Pipe Attraction Zone. Typically, the cost factor value is a number less than one. The lower the number, the more the zone attracts pipe.

Pipe Avoidance Zone

The Pipe Avoidance Zone models a volume that the pipe can enter if it is essential, but will avoid if possible. For example, it is not good practice for pipe to be placed within the immediate vicinity of a tank, but some of the pipes must connect to nozzles on the tank.

Cost Factor

Specifies the cost penalty applied to pipe that runs in the Pipe Avoidance Zone. Typically, the cost factor value is a number greater than one. The higher the number, the more the zone repels pipe.

Pipe Connection Zone

Pipe Connection Zones define a volume at which a route can terminate. They may act as obstruction zones (as defined by the **Obstruction** property) in addition to their special connection properties.

Connection Face

Specifies on which face the connection is made.

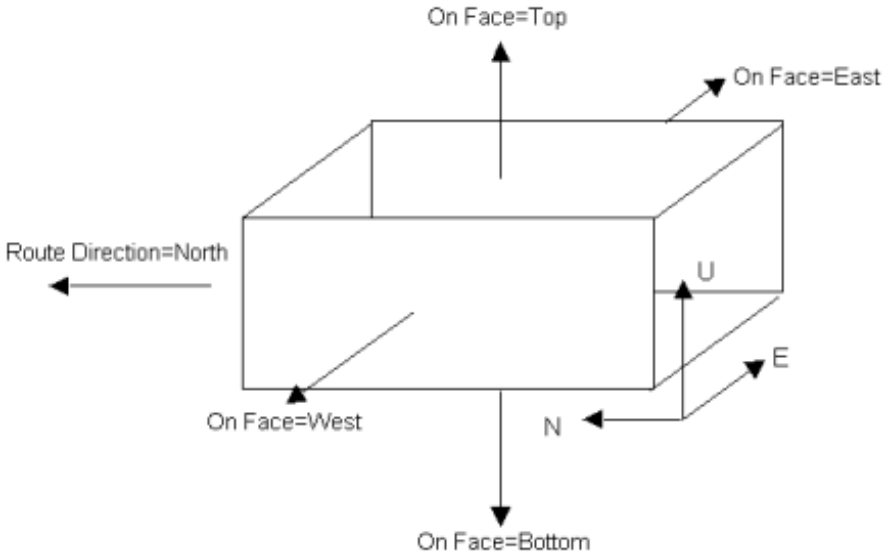
Obstruction

Specifies whether the zone allows pipes to be routed through it (value = **False**) or whether the zone allows pipes to be routed up to it, but not through it (value = **True**).

Pipe Rack Zone

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or

non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Layout

Specifies how pipes are laid out. The software supports five layout types:

At Edge - Pipes are laid out in joining order starting at each edge and working inwards.

Large Bore At Edge - Pipes are laid out in bore order starting at each edge and working inwards.

Spreadout - Pipes are laid out evenly across the rack.

At Center - Pipes are laid out in joining order starting at the center of the rack and working outwards.

Large Bore At Center - Pipes are laid out with the largest bore at the center and then working outwards.

Allow

Filters the pipes that can use the zone based on fluid type. Only the fluid types you specify are permitted on the rack. If more than one fluid type is to be allowed, use a comma or a space to separate them. For example, type **Process,Steam**.

Cost Factor

Displays the cost factor applied to pipe that runs on the rack. The software calculates this value based on data you specify for properties in the **Cost Estimation** category. Typically, the cost factor value is a number less than one. The lower the number, the more the zone attracts pipe.

Custom

Displays all custom properties defined for the volume in the reference data. You can change any of the values on this dialog box both before and after placement.

User Attribute

Displays the name of the custom property defined in the reference data. All custom attributes for the selected volume object appear.

Value

Displays the values for the custom properties of the selected volume object. You can change these values as needed.

IMPORTANT The following category is only displayed when you select either **Pipe Rack Zone** or **Icarus Open Steel Zone** in the **Type** list.

Cost Estimation

Displays properties required by Aspen Icarus Process Evaluator™ to compute a cost that is based on Alias auto- routing.

NOTE Aspen Icarus Process Evaluator is a third-party software product that allows Process Engineers to accurately evaluate the economic impact of their process designs.

Length

Specify the cost estimation length.

Width

Specify the cost estimation width.

Height

Specify the cost estimation height.

Number of Stairways

Specify the number of stairways for cost estimation.

Grating Type

Specify the type of grating used in the zone for cost estimation.

Structural Steel Analysis

Specify the type of structural steel analysis for cost estimation.

Column Base End Condition

Specify the type of connection at the column base for cost estimation.

Distributed Load Per Level

Specify the distributed load per level for cost estimation.

Wind Force Adjustment

Specify the adjustment for wind force for cost estimation.

Seismic Force Adjustment

Specify the adjustment for seismic force for cost estimation.

★IMPORTANT The following **Cost Estimation** properties are only available when you select **Pipe Rack Zone** in the **Type** list.

Pipe Rack Type

Specify the type of pipe rack for **Pipe Rack Zone** cost estimation.

Concrete Type

Specify the type of concrete for **Pipe Rack Zone** cost estimation.

Number of Levels

Specify the number of levels for **Pipe Rack Zone** cost estimation.

Height First Level

Specify the height of the first level for **Pipe Rack Zone** cost estimation. The default measurement is in feet and inches.

Number of Ladders

Specify the number of ladders for **Pipe Rack Zone** cost estimation.

Number of Braced Bays

Specify the number of braced bays for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Pipe Level

Specify the number of beam struts on each pipe level for **Pipe Rack Zone** cost estimation.

Number Catwalks

Specify the number of catwalks for **Pipe Rack Zone** cost estimation.

Catwalk Width

Specify the required width of the catwalk for **Pipe Rack Zone** cost estimation.

Main Bent Spacing

Specify the main bent spacing for **Pipe Rack Zone** cost estimation.

Third Column Option

Specify whether or not a third column is necessary for **Pipe Rack Zone** cost estimation. For a two-bay rack, a third column is required.

Air Cooler Loading

Specify the maximum load for the air cooler for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Column Line

Specify the number of beam struts for each column line for **Pipe Rack Zone** cost estimation.

End Bent Exclusion

Specify whether or not to excluded end bents in the zone for **Pipe Rack Zone** cost estimation.

Minimum Beam Or Column Width

Specify the minimum width of the beam or column for **Pipe Rack Zone** cost estimation.

★ IMPORTANT The following **Cost Estimation** properties are only available when you select **Icarus Open Steel Zone** in the **Type** list.

Number of Floors

Specify the number of floors for **Icarus Open Steel Zone** cost estimation.

Bay Span

Specify the span of the bay for **Icarus Open Steel Zone** cost estimation.

Bay Width

Specify the width of the bay for **Icarus Open Steel Zone** cost estimation.

Floor Grate Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Siding Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Floor Slab Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Slab Thickness

Specify the thickness of the slab for **Icarus Open Steel Zone** cost estimation.

See Also
Ship Zone Properties Dialog Box (on page 138)

Void Space Properties Dialog Box

Displays void space properties for review and editing.

NOTE The **Cross-Section** tab only appears for void spaces created with the **Place Volume Along Path** command.

General Tab (Void Space Properties Dialog Box) (on page 146)
Cross-Section Tab (on page 137)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Void Space Properties Dialog Box)

Displays and defines the general properties of the selected void space.

NOTE Because void space properties are customizable in the reference data, only the properties that are required by the software are documented.

Void space properties are divided into several different categories: **Standard**, **Surface Treatment and Coating**, and **Dimensions**. You select which category that you want to define values for by using the **Category** option.

Type

Displays the type of volume. Selecting **More** from the drop-down list opens the *Select Volume Dialog Box* (on page 29) from which you can select a type.

Standard

Name

Displays or defines the name of the void space. The name is based on the **Name Rule** selection. If you want to type a new name, in the **Name Rule** box, select **User Defined**, and then type a name in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this void space. You can select one of the listed rules or select **User Defined** to specify the name yourself in the **Name** box.

Space Folder

Selects the space folder in which the selected void space resides. Space folders are used to organize volume objects within the model.

Class

Displays the classification of the volume as a compartment or zone. This field is read-only and updates automatically when you select a type of volume in the **Type** box.

Surface Treatment and Coating

Reference Position

Displays the position of the void space with reference to the ship.

Dimensions

Center of Gravity X

Displays the X-coordinate for the center of gravity of the volume.

Center of Gravity Y

Displays the Y-coordinate for the center of gravity of the volume.

Center of Gravity Z

Displays the Z-coordinate for the center of gravity of the volume.

Volume

Specifies the volume of the compartment or zone.

Moment of Inertia X

Displays the X-coordinate for the moment of inertia of the volume.

Moment of Inertia Y

Displays the Y-coordinate for the moment of inertia of the volume.

Moment of Inertia Z

Specifies the Z-coordinate for the moment of inertia of the volume.

Surface Area

Specifies the surface area value of the volume.

Range High X

Displays the X-coordinate for the upper end of the volume.

Range High Y

Displays the Y-coordinate for the upper end of the volume.

Range High Z

Displays the Z-coordinate for the upper end of the volume.

Range Low X

Displays the X-coordinate for the lower end of the volume.

Range Low Y

Displays the Y-coordinate for the lower end of the volume.

Range Low Z

Displays the Z-coordinate for the lower end of the volume.

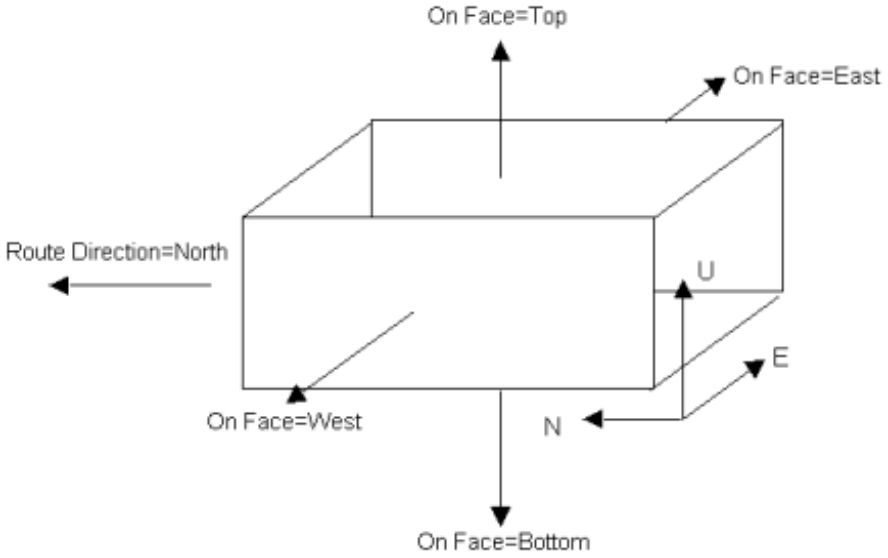
★ IMPORTANT The following route zones will display additional **Standard** category properties specific to their zone type.

Pipe Attraction Zone

The Pipe Attraction Zone models a volume that attracts pipes. It has a cost factor, which means that routes that lie within the Pipe Attraction Zone are less expensive than equivalent routes outside the zone. The Pipe Attraction Zone is similar to the Pipe Rack Zone in that it modifies the result of the routing calculation by attracting pipes but has fewer rules for layout and for joining and leaving the zone.

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always

run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Cost Factor

Specifies the cost factor applied to pipe that runs through the Pipe Attraction Zone. Typically, the cost factor value is a number less than one. The lower the number, the more the zone attracts pipe.

Pipe Avoidance Zone

The Pipe Avoidance Zone models a volume that the pipe can enter if it is essential, but will avoid if possible. For example, it is not good practice for pipe to be placed within the immediate vicinity of a tank, but some of the pipes must connect to nozzles on the tank.

Cost Factor

Specifies the cost penalty applied to pipe that runs in the Pipe Avoidance Zone. Typically, the cost factor value is a number greater than one. The higher the number, the more the zone repels pipe.

Pipe Connection Zone

Pipe Connection Zones define a volume at which a route can terminate. They may act as obstruction zones (as defined by the **Obstruction** property) in addition to their special connection properties.

Connection Face

Specifies on which face the connection is made.

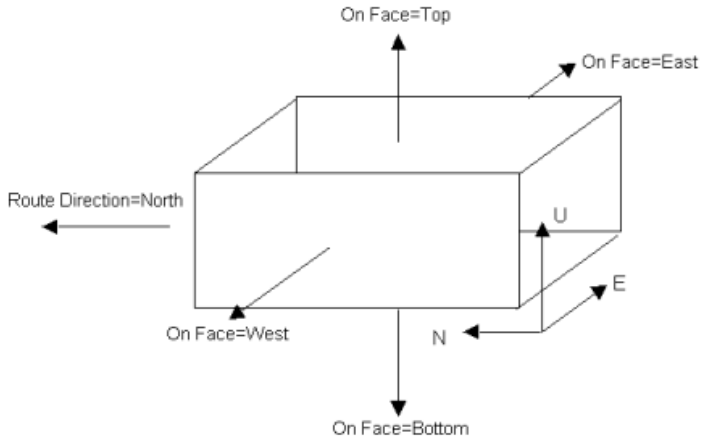
Obstruction

Specifies whether the zone allows pipes to be routed through it (value = **False**) or whether the zone allows pipes to be routed up to it, but not through it (value = **True**).

Pipe Rack Zone

On Face

Defines on which of the six faces the pipe is to be routed as shown in the following illustration:



In a Pipe Attraction Zone, the **On Face** property is not always obeyed if doing so will result in a change in the pipe routing. In other words, the software will not add bends to the route in order to force a pipe to run on the specified face. This behavior is in contrast to behavior of the **On Face** property in a Pipe Rack Zone. As such, Intergraph recommends that the **On Face** property always be defined for a Pipe Attraction Zone in order to avoid unpredictable behavior. If the property is left undefined, the zone will select any one of the four candidate faces as an appropriate default based on the **Route Direction**. For example, if the **Route Direction** is set to East, the software can choose Top, Bottom, North, or South faces as the default, whereas setting the **Route Direction** to West can produce a different result.

Associated

Determines whether all pipes see the zone as a candidate for routing, or only those pipes that explicitly reference the zone. If the value of the property is set to **True**, only those pipes with the zone in their list of zones to use will do so. If the value of the property is set to **False**, all pipes will evaluate the zone for use in routing.

Route Direction

Specifies the direction relative to the local coordinate system in which pipe will be routed. The **Route Direction** property defines the axis of the routed pipe, so an **East** value is the same as a **West** value, **North** is equal to **South**, and **Up** is equal to **Down**. It is unrelated to the direction of the pipe as it is routed between its From and To connections. For example, if a pipe is routed from West to East through a pipe attraction zone or rack, the result is the same whether the **Route Direction** property is set to East or West. Because pipes always run parallel to one of the pipe attraction zone or rack axes, the route direction in a skewed or non-orthogonal rack is translated as the axis closest to the specified orthogonal axis. If a rack is at 45 degrees to orthogonal, the software uses the longer of the two candidate axes.

Layout

Specifies how pipes are laid out. The software supports five layout types:

At Edge - Pipes are laid out in joining order starting at each edge and working inwards.

Large Bore At Edge - Pipes are laid out in bore order starting at each edge and working inwards.

Spreadout - Pipes are laid out evenly across the rack.

At Center - Pipes are laid out in joining order starting at the center of the rack and working outwards.

Large Bore At Center - Pipes are laid out with the largest bore at the center and then working outwards.

Allow

Filters the pipes that can use the zone based on fluid type. Only the fluid types you specify are permitted on the rack. If more than one fluid type is to be allowed, use a comma or a space to separate them. For example, type **Process,Steam**.

Cost Factor

Displays the cost factor applied to pipe that runs on the rack. The software calculates this value based on data you specify for properties in the **Cost Estimation** category. Typically, the cost factor value is a number less than one. The lower the number, the more the zone attracts pipe.

Custom

Displays all custom properties defined for the volume in the reference data. You can change any of the values on this dialog box both before and after placement.

User Attribute

Displays the name of the custom property defined in the reference data. All custom attributes for the selected volume object appear.

Value

Displays the values for the custom properties of the selected volume object. You can change these values as needed.

IMPORTANT The following category is only displayed when you select either **Pipe Rack Zone** or **Icarus Open Steel Zone** in the **Type** list.

Cost Estimation

Displays properties required by Aspen Icarus Process Evaluator™ to compute a cost that is based on Alias auto- routing.

NOTE Aspen Icarus Process Evaluator is a third-party software product that allows Process Engineers to accurately evaluate the economic impact of their process designs.

Length

Specify the length.

Width

Specify the width.

Height

Specify the height.

Number of Stairways

Specify the number of stairways.

Grating Type

Specify the type of grating used in the zone.

Structural Steel Analysis

Specify the type of structural steel analysis.

Column Base End Condition

Specify the type of connection at the column base.

Distributed Load Per Level

Specify the distributed load per level.

Wind Force Adjustment

Specify the adjustment for wind force.

Seismic Force Adjustment

Specify the adjustment for seismic force.

★IMPORTANT The following **Cost Estimation** properties are only available when you select **Pipe Rack Zone** in the **Type** list.

Pipe Rack Type

Specify the type of pipe rack for **Pipe Rack Zone** cost estimation.

Concrete Type

Specify the type of concrete for **Pipe Rack Zone** cost estimation.

Number of Levels

Specify the number of levels for **Pipe Rack Zone** cost estimation.

Height First Level

Specify the height of the first level for **Pipe Rack Zone** cost estimation. The default measurement is in feet and inches.

Number of Ladders

Specify the number of ladders for **Pipe Rack Zone** cost estimation.

Number of Braced Bays

Specify the number of braced bays for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Pipe Level

Specify the number of beam struts on each pipe level for **Pipe Rack Zone** cost estimation.

Number Catwalks

Specify the number of catwalks for **Pipe Rack Zone** cost estimation.

Catwalk Width

Specify the required width of the catwalk for **Pipe Rack Zone** cost estimation.

Main Bent Spacing

Specify the main bent spacing for **Pipe Rack Zone** cost estimation.

Third Column Option

Specify whether or not a third column is necessary for **Pipe Rack Zone** cost estimation. For a two-bay rack, a third column is required.

Air Cooler Loading

Specify the maximum load for the air cooler for **Pipe Rack Zone** cost estimation.

Number of Beam Struts Per Column Line

Specify the number of beam struts for each column line for **Pipe Rack Zone** cost estimation.

End Bent Exclusion

Specify whether or not to excluded end bents in the zone for **Pipe Rack Zone** cost estimation.

Minimum Beam Or Column Width

Specify the minimum width of the beam or column for **Pipe Rack Zone** cost estimation.

IMPORTANT The following **Cost Estimation** properties are only available when you select **Icarus Open Steel Zone** in the **Type** list.

Number of Floors

Specify the number of floors for **Icarus Open Steel Zone** cost estimation.

Bay Span

Specify the span of the bay for **Icarus Open Steel Zone** cost estimation.

Bay Width

Specify the width of the bay for **Icarus Open Steel Zone** cost estimation.

Floor Grate Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Siding Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Floor Slab Percent Area

Specify the percent of the area through which you can route pipe for **Icarus Open Steel Zone** cost estimation.

Slab Thickness

Specify the thickness of the slab for **Icarus Open Steel Zone** cost estimation.

See Also

Void Space Properties Dialog Box (on page 146)

Sketch Properties Dialog Box

Allows you to view information about the points in a path. You cannot edit the data on this dialog box.

General Tab (Sketch Properties Dialog Box) (on page 153)

General Tab (Sketch Properties Dialog Box)

Displays the points that make up a path, their coordinates, turn types, and turn type dimensions.

Point No

Displays the point number that identifies the selected point.

X

Displays the location of the point on the X-axis.

Y

Displays the location of the point on the Y-axis.

Z

Displays the location of the point on the Z-axis.

Turn Type

Displays the type of turn associated with the point. Turn types include none, bend, chamfer, and cornice.

Value

Specifies dimensions for the selected turn type. For bends and cornices, the value specifies the radius of the bend. For chamfers, the value specifies the dimensions for setback A and setback B of the chamfer.

See Also

Sketch Properties Dialog Box (on page 153)

Space Folder Properties Dialog Box

Displays space folder properties for review and editing.

General Tab (Space Folder Properties Dialog Box) (on page 154)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Space Folder Properties Dialog Box)

Displays and defines the general properties of the selected space folder.

Standard

Name

Displays or defines the name of the space folder. You can type a different name.

Space Folder

Specifies the parent to which the selected folder belongs. The parent determines the placement of the folder within the space hierarchy. Selecting **More** in the drop-down list displays the *Select Space Folder Dialog Box* (on page 17).

See Also

Space Folder Properties Dialog Box (on page 154)

Spatial Load Folder Properties Dialog Box

Displays load folder properties for review and editing.

General Tab (Load Folder Properties Dialog Box) (on page 155)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Load Folder Properties Dialog Box)

Sets general options for the selected load folder.

Standard

Name

Displays or defines the name of the load folder. You can type a different name.

Parent

Specifies the parent to which the selected load folder belongs. The parent determines the placement of the folder within the load hierarchy. Selecting **More** in the drop-down list displays the *Select Load Folder Dialog Box* (on page 96).

See Also

Spatial Load Folder Properties Dialog Box (on page 154)

Spatial Load Combination Properties Dialog Box

Displays load combination properties for review and editing.

General Tab (Spatial Load Combination Properties Dialog Box) (on page 155)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Spatial Load Combination Properties Dialog Box)

Category

Load combination properties are divided into different categories: **Standard** and **Weight and CG**. You can select the category to define values for by using the **Category** option.

Standard

Name

Specifies the name of the load combination.

Name Rule

Select a name rule to use to name the load combination. Select **User Defined** if you want to name the load combination yourself.

Parent

Specifies the parent to which the selected load combination belongs. The parent determines the placement of the load combination within the load hierarchy. Selecting **More** in the drop-down list displays the *Select Load Folder Dialog Box* (on page 96).

Sea Pressure

Select the sea pressure value for the load combination. If you want to add, edit, or remove values that are available for selection, edit the **Load Case Sea Pressure** sheet in the

CompartmentCodeLists.xls workbook in the reference data.

Liquid Pressure

Select the liquid pressure value for the load combination. If you want to add, edit, or remove values that are available for selection, edit the **Load Case Liquid Pressure** sheet in the **CompartmentCodeLists.xls** workbook in the reference data.

Weight and CG

Dry Weight

Displays the dry weight of the object.

Wet Weight

Displays the wet weight of the object.

Dry CG X

Displays the X-coordinate of the center-of-gravity for the dry object.

Dry CG Y

Displays the Y-coordinate of the center-of-gravity for the dry object.

Dry CG Z

Displays the Z-coordinate of the center-of-gravity for the dry object.

Wet CG X

Displays the X-coordinate of the center-of-gravity for the wet object.

Wet CG Y

Displays the Y-coordinate of the center-of-gravity for the wet object.

Wet CG Z

Displays the Z-coordinate of the center-of-gravity for the wet object.

See Also

Spatial Load Combination Properties Dialog Box (on page 155)

Spatial Load Properties Dialog Box

Displays spatial load properties for review and editing.

General Tab (Spatial Load Properties Dialog Box) (on page 157)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Spatial Load Properties Dialog Box)

Category

Load properties are divided into different categories: **Standard** and **Weight and CG**. You can select the category to define values for by using the **Category** option.

Standard

Name

Specifies the name of the load.

Name Rule

Select a name rule to use to name the load. Select **User Defined** if you want to name the load yourself.

Parent

Specifies the parent to which the selected load belongs. The parent determines the placement of the load within the load hierarchy. Selecting **More** in the dropdown list displays the *Select Load Folder Dialog Box* (on page 96).

NOTE The other properties in the **Standard** category depend on the type of material in the cargo compartment.

Weight and CG

Dry Weight

Displays the dry weight of the object.

Wet Weight

Displays the wet weight of the object.

Dry CG X

Displays the X-coordinate of the center-of-gravity for the dry object.

Dry CG Y

Displays the Y-coordinate of the center-of-gravity for the dry object.

Dry CG Z

Displays the Z-coordinate of the center-of-gravity for the dry object.

Wet CG X

Displays the X-coordinate of the center-of-gravity for the wet object.

Wet CG Y

Displays the Y-coordinate of the center-of-gravity for the wet object.

Wet CG Z

Displays the Z-coordinate of the center-of-gravity for the wet object.

See Also

Spatial Load Properties Dialog Box (on page 156)

Unit Load Properties Dialog Box

Displays unit load properties for review and editing.

General Tab (Unit Load Properties Dialog Box) (on page 158)
Relationship Tab (on page 125)
Configuration Tab (on page 124)
Notes Tab (on page 125)

General Tab (Unit Load Properties Dialog Box)

Category

Load properties are divided into different categories: **Standard** and **Weight and CG**. You can select the category to define values for by using the **Category** option.

Standard

Name

Specifies the name of the load.

Name Rule

Select a name rule to use to name the load. Select **User Defined** if you want to name the load yourself.

Parent

Specifies the parent to which the selected load belongs. The parent determines the placement of the load within the load hierarchy. Selecting **More** in the drop-down list displays the *Select Load Folder Dialog Box* (on page 96).

Magnitude

Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Direction** box.

Direction

Select the direction in which the load magnitude is to be applied. You can place loads in any of the six directions: Forward, Aft, Port Side, Starboard, Up, Down, or User Defined.

X

Specifies the X-coordinate of the unit vector representing the force of the load.

Y

Specifies the Y-coordinate of the unit vector representing the force of the load.

Z

Specifies the Z-coordinate of the unit vector representing the force of the load.

XPosition

Specifies the X-coordinate for the position of the load.

YPosition

Specifies the Y-coordinate for the position of the load.

ZPosition

Specifies the Z-coordinate for the position of the load.

Permeability

Specifies the permeability of the cargo compartment.

Stowage factor

Specifies the stowage factor of the cargo compartment.

Volume

Specifies the volume of the cargo compartment.

Stack limit

Specifies a stack limit value for the cargo compartment.

Weight

Specifies the weight of the cargo compartment.

Weight and CG

Dry Weight

Displays the dry weight of the object.

Wet Weight

Displays the wet weight of the object.

Dry CG X

Displays the X-coordinate of the center-of-gravity for the dry object.

Dry CG Y

Displays the Y-coordinate of the center-of-gravity for the dry object.

Dry CG Z

Displays the Z-coordinate of the center-of-gravity for the dry object.

Wet CG X

Displays the X-coordinate of the center-of-gravity for the wet object.

Wet CG Y

Displays the Y-coordinate of the center-of-gravity for the wet object.

Wet CG Z

Displays the Z-coordinate of the center-of-gravity for the wet object.

See Also

Unit Load Properties Dialog Box (on page 158)

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

built ships

Complete database of NGC information after completion of the ship contract.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the

case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

Index

D

E

F

G

H

I

Q

R

S

T

U

V

W